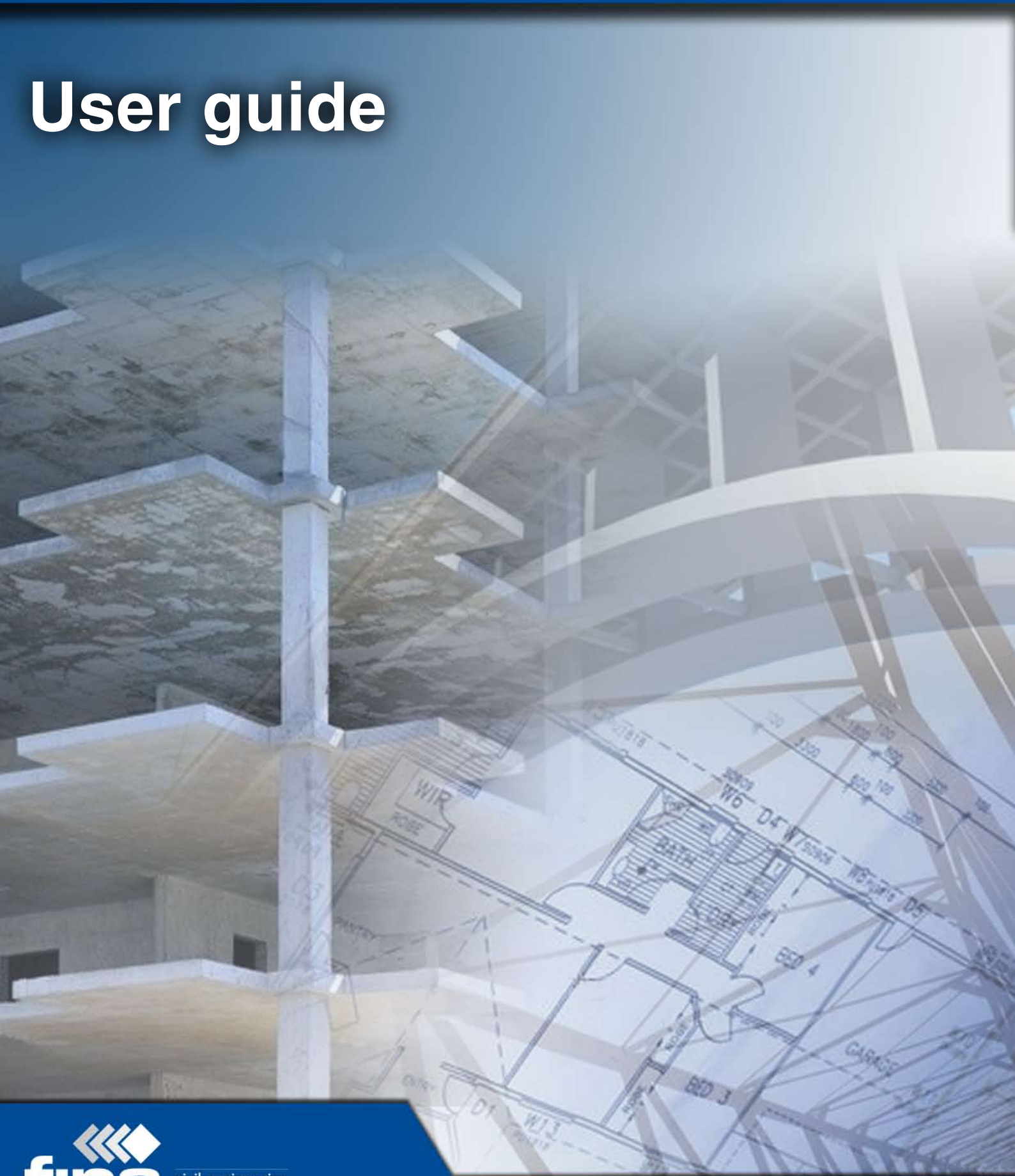
structural engineering

FIN EC

User guide



fine civil engineering software

www.finesoftware.eu

Content

Introduction.....3
RC column.....4
Punching shear.....21
Timber truss.....33
3D structure.....64

Introduction

This User Manual is intended for all users of the Fin EC structural analysis programs. Simple examples are used to explain the principles of the basic structural analysis programs (Fin 2D and Fin 3D) as well as of the design programs for concrete, steel, timber and masonry structures.
The examples show procedures for analyses of particular tasks in selected programs. Due to similar principles of work, each example can be used as guidance for other programs of the same type. For instance, the example "**Reinforced concrete column**" shows work with the "**Concrete 3D**", however the same principles of quick design of a section subject to a combination of internal forces can be applied in all other design programs for concrete, steel, timber and masonry structures.
The User Manual contains the following examples:

Reinforced concrete column
This example shows design of a reinforced concrete cross-section subject to given combination of internal forces in the program "**Concrete 3D**". The same principles of design are used in all other design programs – "**Concrete 2D**", "**Plain concrete**", "**Steel**", "**Steel Fire**", "**Timber**", "**Timber Fire**" and "**Masonry**". The example also shows the procedure for composition of output documentation.

Punching shear
In this example, work procedure in the program "**Punching**" is explained. Apart from that, the procedure for editing headers of output documents, which can also be applied in other programs Fin EC, is described.

Timber truss
The example shows the procedure for modelling a timber truss using the generator of structures in the program "**Fin 2D**" and subsequent design of members in the program "**Timber**". It also describes utilization of design groups and collective definition of calculation parameters in the design program. This example is also suitable for users of the program "**Fin 3D**".

3D structure
This example shows analysis of a 3D structure in the program "**Fin 3D**" and subsequent design of members in the program "**Steel**". Apart from gradual definition of the structure in the program "**Fin 3D**" using joints and members, work with the design groups is described. These procedures can also be used in the program "**Fin 2D**" in connection with any of the design programs.

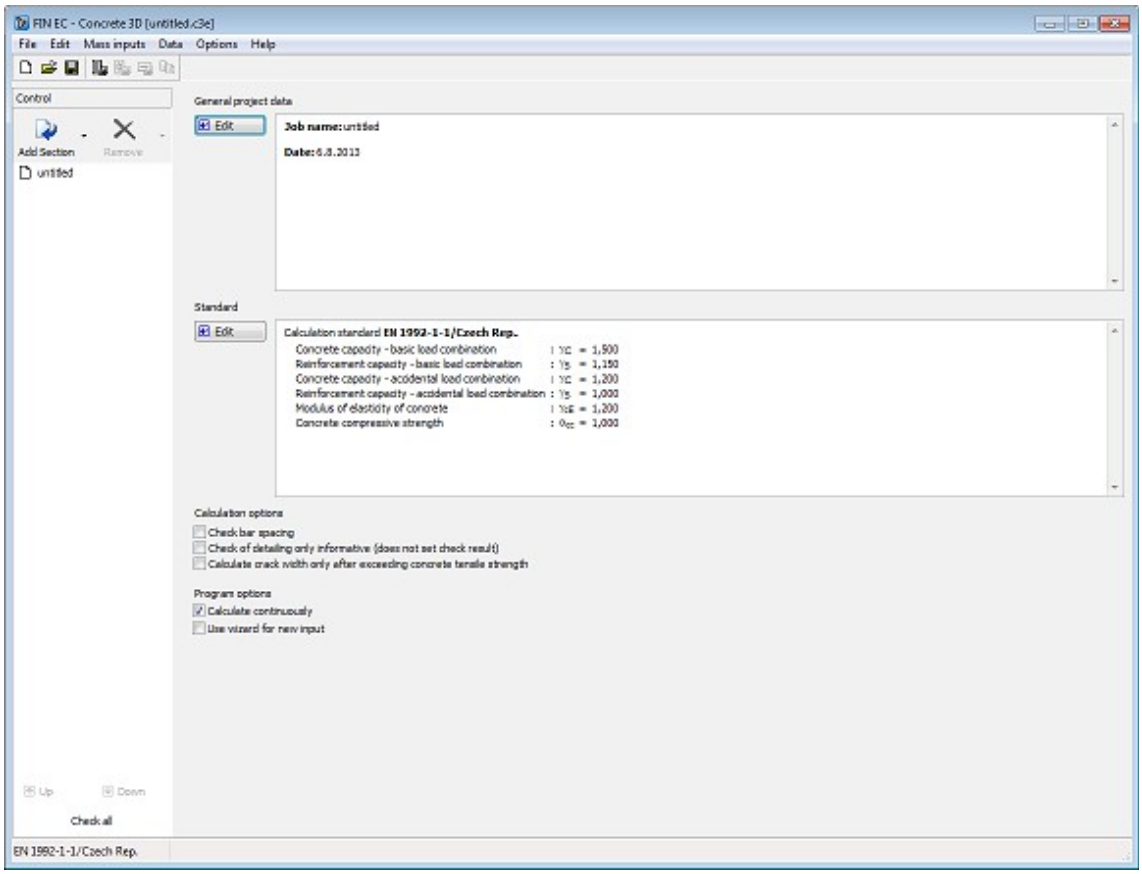
RC column

Task

In this example, the aim is to design reinforcement of a concrete column of a hexagonal cross-section. With *200mm* depth of the section and *2000mm* length, the column is subject to axial compression and biaxial bending. The actions in the ultimate limit state are: N_x= *400kN*, M_y= *2.33kNm* and M_z= *5.46kNm*; in serviceability limit state: N_x= *350kN* and M_y= *2.00kNm*. Concrete of strength class *C30/37 X0* and steel of grade *B500* are used in the design.

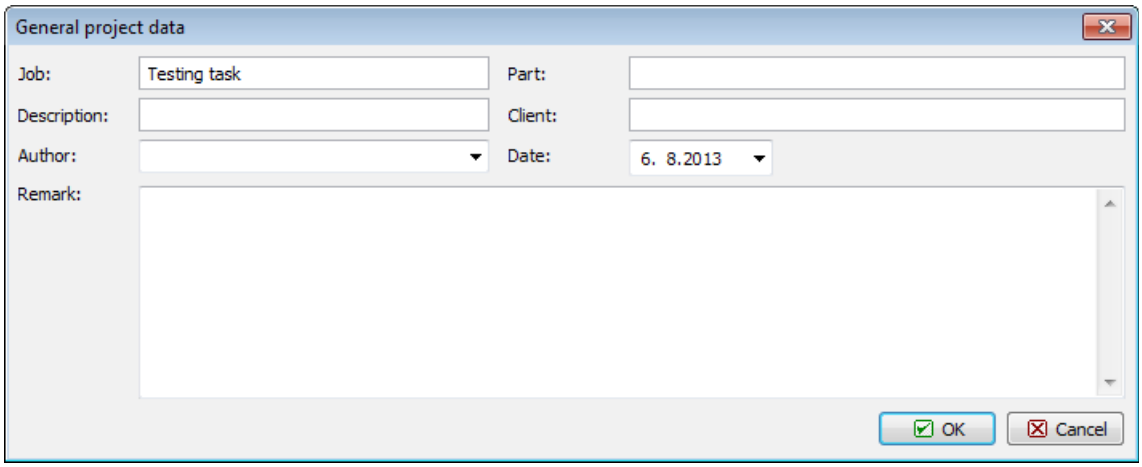
Creating a new job

The following screen appears after running program "**Concrete 3D**":


FIN EC - Concrete 3D [untitled.c3e]
File Edit Mass inputs Data Options Help
Control
Add Section
Remove
untitled
General project data
Edit
Job name: untitled
Date: 6.8.2013
Standard
Edit
Calculation standard EN 1992-1-1/Czech Rep.
Concrete capacity - basic load combination : γc = 1,500
Reinforcement capacity - basic load combination : γs = 1,150
Concrete capacity - accidental load combination : γc = 1,200
Reinforcement capacity - accidental load combination : γs = 1,000
Modulus of elasticity of concrete : γE = 1,200
Concrete compressive strength : αcc = 1,000
Calculation options
Check bar spacing
Check of detailing only informative (does not set check result)
Calculate crack width only after exceeding concrete tensile strength
Program options
Calculate continuously
Use wizard for new input
Up
Down
Check all
EN 1992-1-1/Czech Rep.

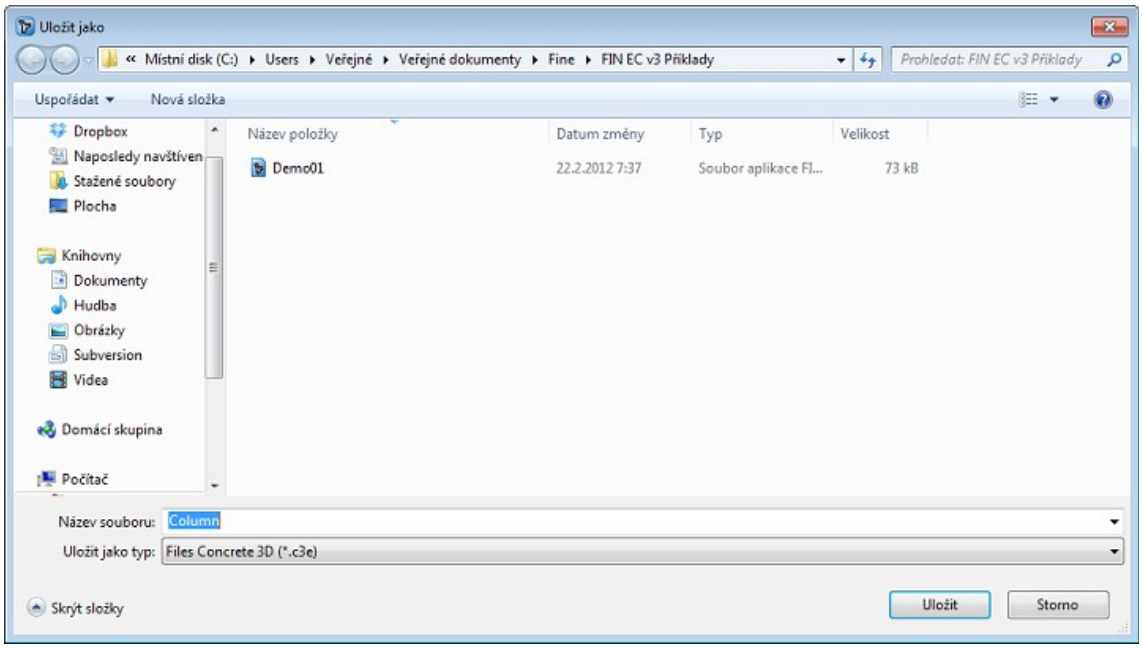
The start screen of the Concrete 3D program

The program enables calculating unlimited number of sections per job, in this example only one section is calculated. In the "**General project data**" section of the start screen, the job name, description and other project identification data can be entered. After clicking the "**Edit**" button, we first enter the job name:



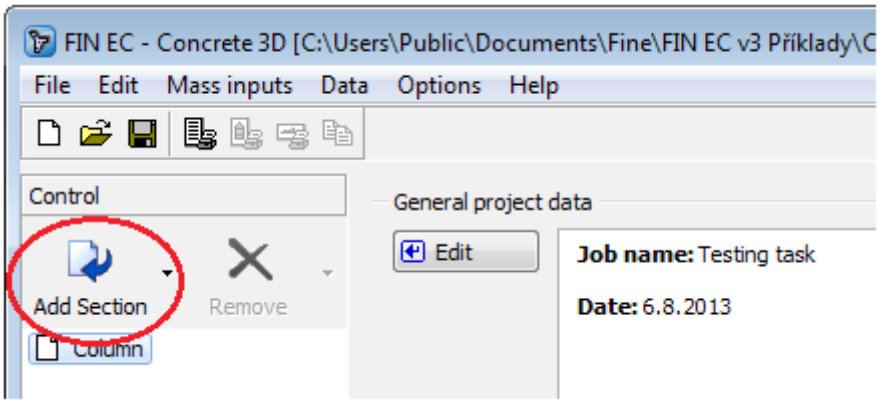
"General project data" dialog box

In the "**General project data**" dialog box, identification data of the job can be entered; these can later be displayed in the header or footer of the output files. In the start screen we can also change settings of some of the calculation parameters and thus affect programs general functionality ("**Calculation options**" and "**Program options**" frames in the lower part of the screen).
Prior to commencing any work, it is advisable to save the job; this can be done either using " " button, or in the main menu clicking on File – Save As, or using the "**Ctrl+S**" shortcut.



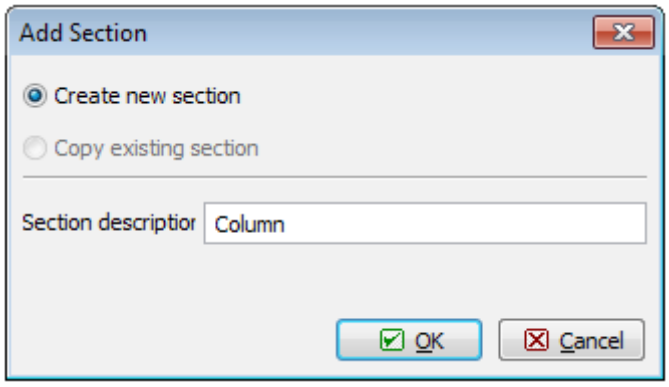
Saving job

Now we can proceed to entering a new section by clicking the "**Add Section**" button in the upper part of the program's control tree on the left-hand side.



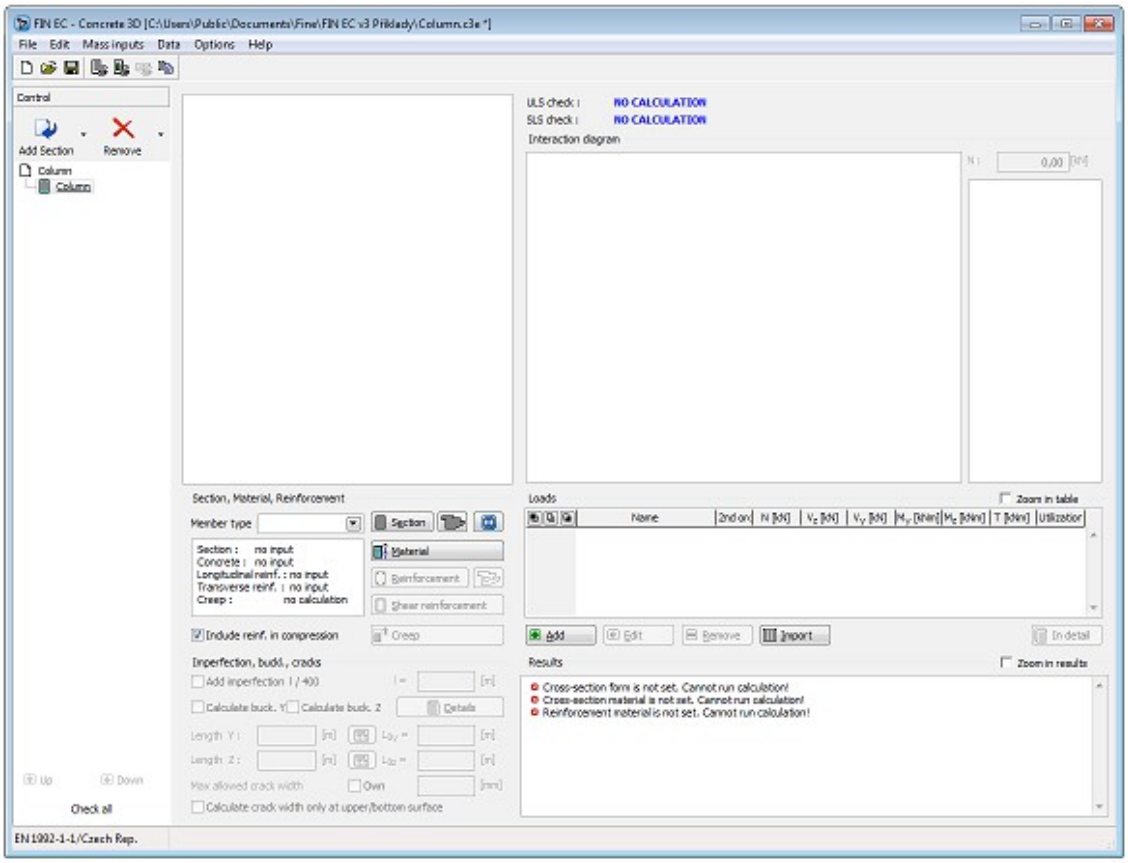
Adding a new section

The following dialog box appears, in which we can enter the section's name ("**Column**") into the "**Section description**" field, confirming by clicking the "**OK**" button.



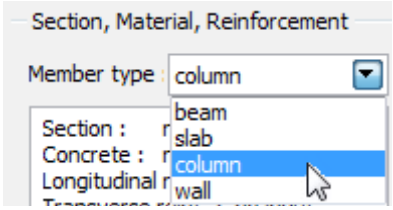
Dialog box for adding a new section

A new item has been generated in the control tree, representing the new added section ("**Column**"). The program has now automatically selected this item; therefore we can directly proceed to entering the section parameters.

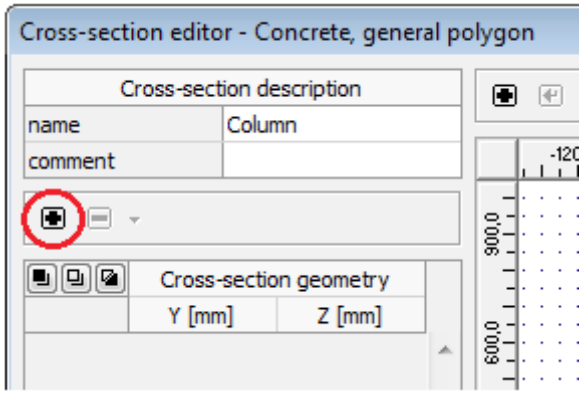


Dialog box for section parameters

Section, Material, Reinforcement
At first it is necessary to enter the basic geometrical and material characteristics of the section in the "**Section**, **Material**, **Reinforcement**" frame. From the "**Member type**" drop-down list we select the function of the member in the structure: **beam**, **slab**, **column** or **wall**.

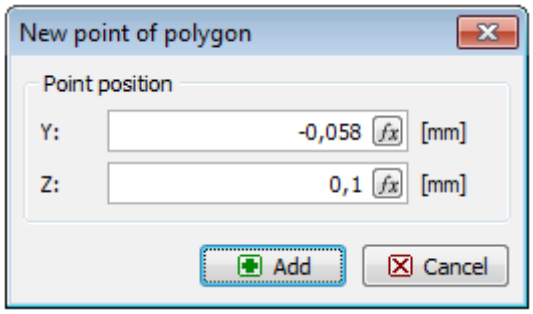


In our example we select member type "**column**". This selection affects the assessment and design checks of the reinforcement arrangement.
As a hexagon is not included in the library of pre-defined cross-sections, we need to use the " " button to define geometry of a general polygon:



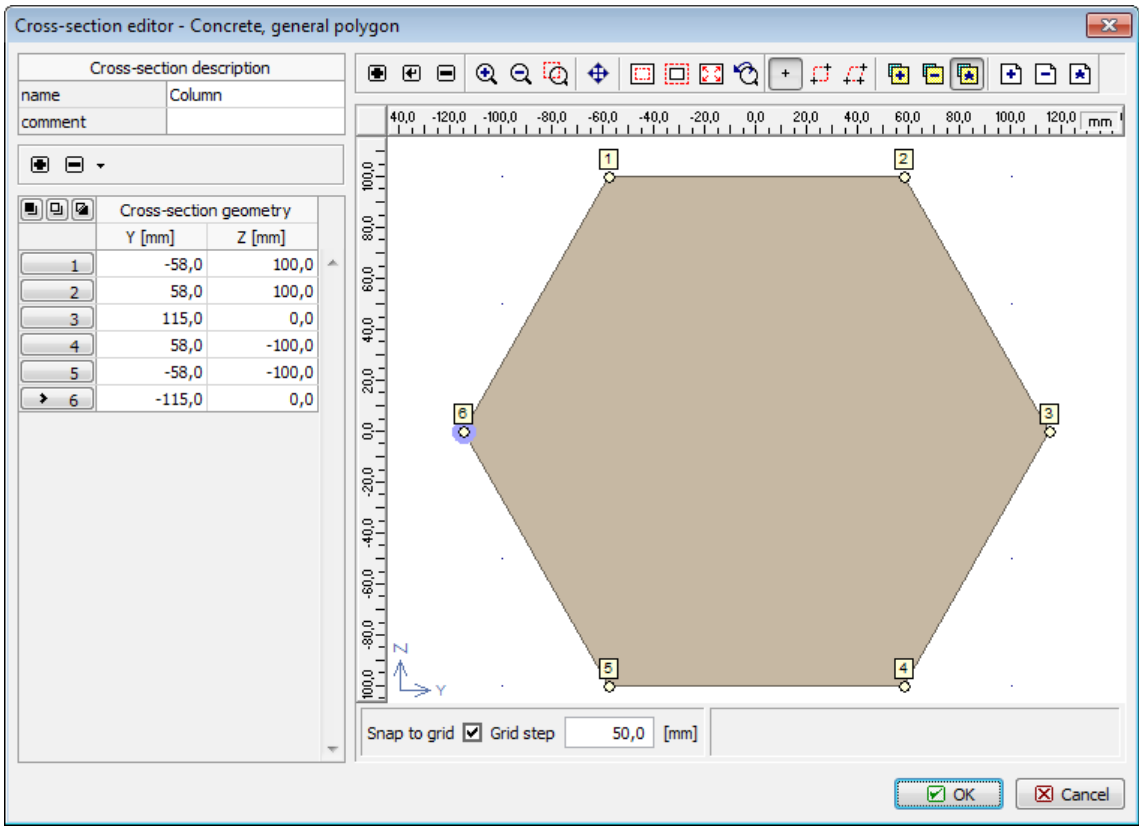
Adding polygon points.

In the dialog box "**Cross-section editor**" we can define geometry of any polygonal cross-section as a set of points. Having the coordinates available, we can define each of the points numerically by clicking the "**+**" button, located in the bar in the left upper corner above the table of points. The following dialog box appears:



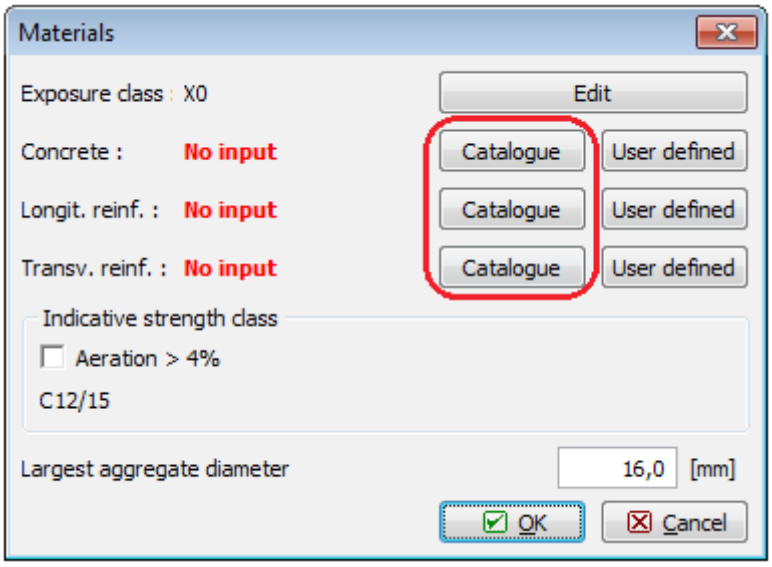
Dialog box for polygon points definition

In this dialog box, one by one we enter the coordinates of all six points of the polygon: *[-0,058;0,100]*, *[0,058;0,100]*, *[0,115;0,000]*, *[0,058;-0,100]*, *[-0,058;-0,100]*, *[-0,115;0,000]*. Each point is confirmed by clicking the "**Add**" button; after entering coordinates of the last point, we return to the "**Cross-section editor**" dialog box by clicking the "**Cancel**" button.
The geometry of the cross-section appears on the right side of the dialog box; we can change the points coordinates either directly in the table on the left side or graphically on the right side. We close the dialog box by clicking the "**OK**" button.



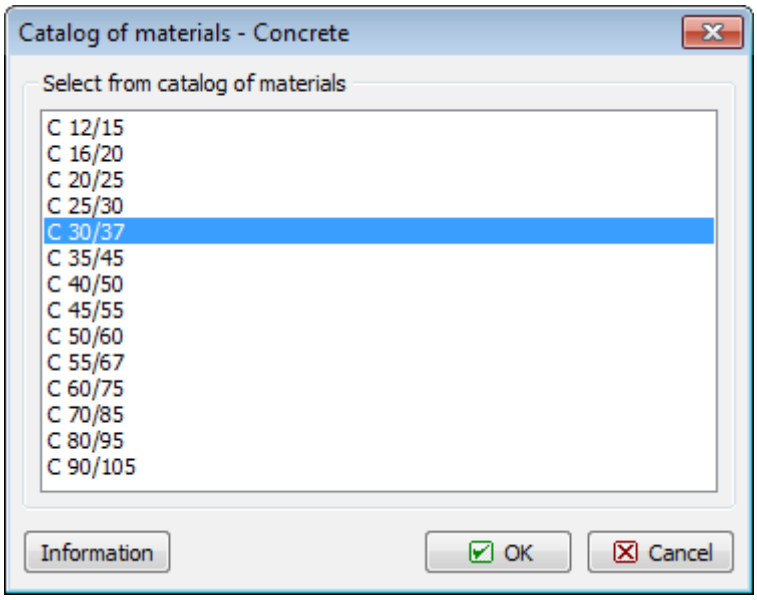
Definition of polygon's geometry

We proceed with defining the material properties in "**Materials**" dialog box which is run by clicking the "**Material**" button in the "**Section**, **Materials**, **Reinforcement**" dialog box. Assuming the column is located inside of the structure we select the "**Exposure class X0 – no risk of corrosion or attack**", as the column is not in contact with outside environment. Subsequently we define the material properties of concrete and longitudinal and transversal reinforcement. We can select standardized materials from the library of pre-defined materials clicking the "**Catalogue**" button at relevant lines.



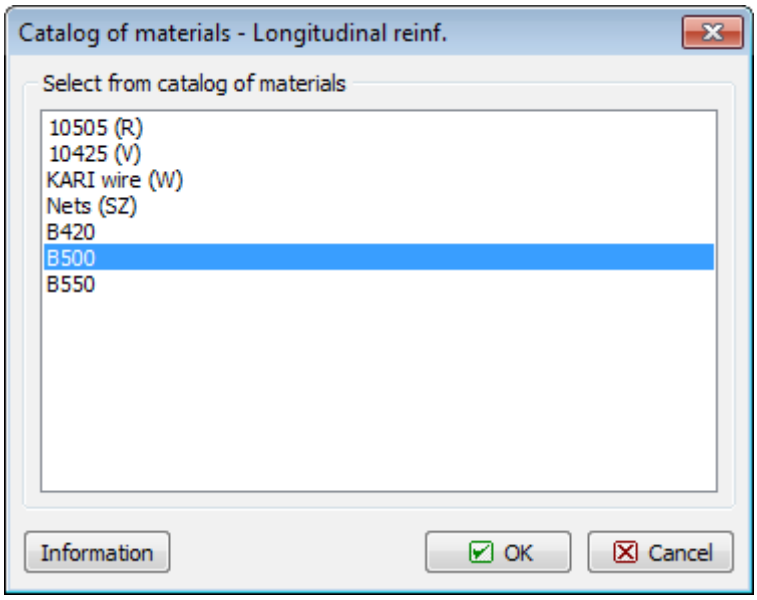
"Materials" dialog box

For concrete we select the strength class "**C30/37**" and close the dialog box by clicking "**OK**".



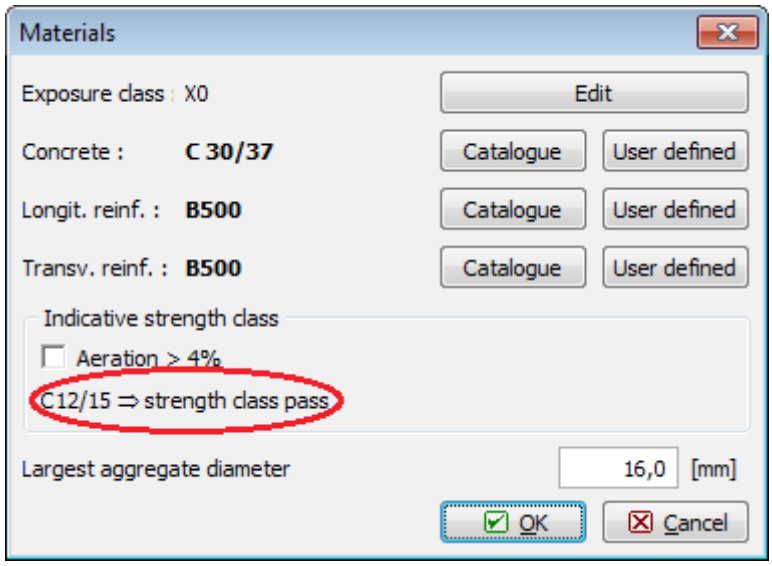
Concrete class selection

Proceeding to definition of the steel properties, we select the grade "**B500**" for both longitudinal and transversal reinforcement and close the dialog box by clicking "**OK**".



Steel grade selection

After returning to the "**Materials**" dialog box we can check the summary of the selected materials and confirm whether the selected class of concrete fulfils requirements on the "**Indicative strength class**" given by the selected Exposure Class. We exit the dialog box by clicking "**OK**".



Indicative strength class check in "Materials" dialog box

Loads

After defining the geometry of the section and material properties, we can proceed either with defining the reinforcement or loads. In our example we first define a load case because then we are able to check the results of the reinforcement assessment during its definition. To create a load case, we click the "**Add**" button located under the "**Loads**" table. In the "**New load**" dialog box we select a "**Combination type**" depending on which combination has been used to obtain the design forces and bending moments and for which design situation the assessment shall be carried out. The following options are available:

Basic design (ULS)
- Forces and bending moments have been obtained from the basic combination for persistent and transient design situations according to EN 1990, equations *6.10* resp. *6.10a* and *6.10b*. These loads are used for basic assessment of cross-section's capacity in the ultimate limit state.

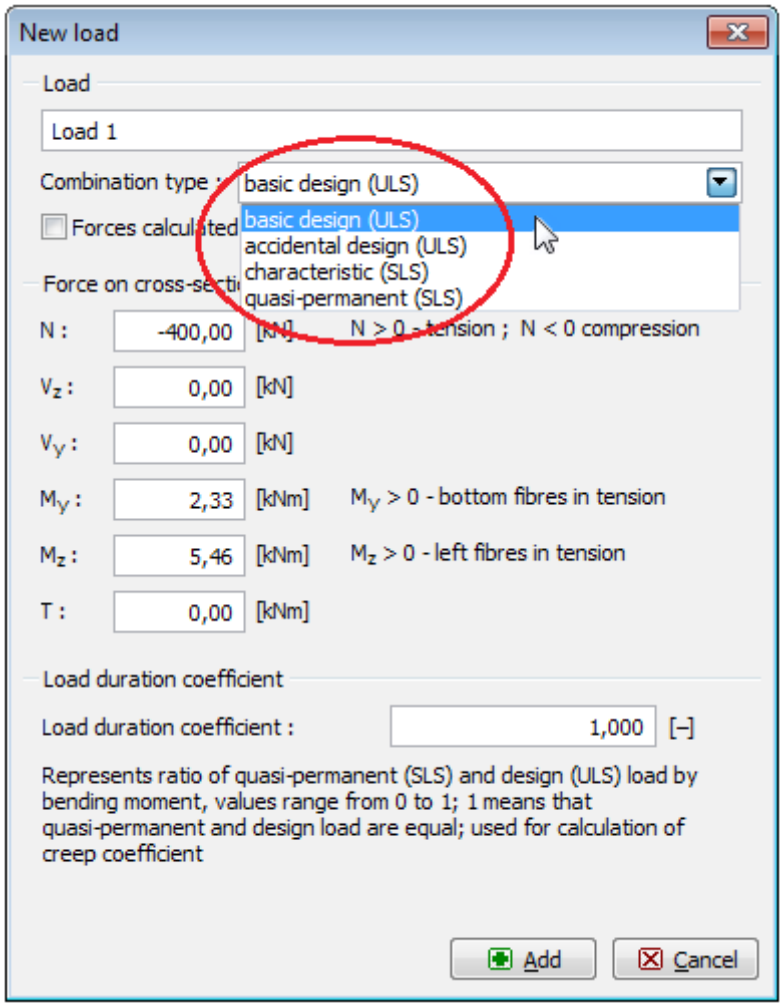
Accidental design (ULS)
- Forces and bending moments have been obtained from the combination for accidental design situations according to EN 1990, equation *6.11*. These loads are used for assessment of cross-section's capacity in accidental design situations in the ultimate limit state (partial safety and material factors for accidental design situations are used).

Characteristic (SLS)
- Forces and bending moments have been obtained from the characteristic combination according to EN 1990, equation *6.14*. These loads are used for assessment of the stress limitation (serviceability limit state).

Quasi-permanent (SLS)
- Forces and bending moments have been obtained from the quasi-permanent combination according to EN 1990, equation *6.16*. These loads are used for assessment of the crack widths in the serviceability limit state.

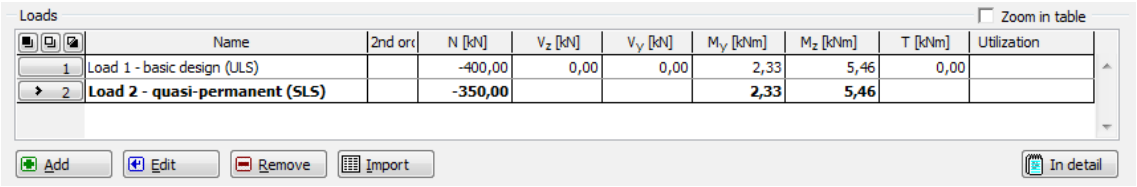
Subsequently we enter the forces and bending moments acting on the cross-section; in our example the axial force is *N= -400kN* (negative value denotes compression) and the bending moments are M_y*= 2,33kNm* and M_z*= 5,46kNm*.

Also, we should enter the "**Load duration coefficient**", i.e. the ratio of quasi-permanent and total loads for calculation of the creep coefficient. If the exact value is not available, we can leave the conservative value of *1.00*, which means that the total load is considered quasi-permanent in the calculations. The new load is confirmed by clicking the "**Add**" button and "**Cancel**" to exit the dialog box.


New load
Load
Load 1
Combination type :
basic design (ULS)
Forces calculated
basic design (ULS)
accidental design (ULS)
characteristic (SLS)
quasi-permanent (SLS)
Force on cross-sec
N : -400,00 [kN] N > 0 - tension ; N < 0 compression
Vz : 0,00 [kN]
Vy : 0,00 [kN]
My : 2,33 [kNm] My > 0 - bottom fibres in tension
Mz : 5,46 [kNm] Mz > 0 - left fibres in tension
T : 0,00 [kNm]
Load duration coefficient
Load duration coefficient : 1,000 [–]
Represents ratio of quasi-permanent (SLS) and design (ULS) load by bending moment, values range from 0 to 1; 1 means that quasi-permanent and design load are equal; used for calculation of creep coefficient
Add
Cancel

New loads definition

To define the loads for the serviceability limit state, analogical procedure is used as for the ultimate limit state loads. To assess the stress limitation, we select "**Combination type – characteristic (SLS)**" and enter the relevant axial compressive force *N= -350kN* and the bending moments M_y*= 2.00kNm* and M_z*= 0.00kNm*.
As result, a table summarizing all defined load cases is generated in the dialog box.



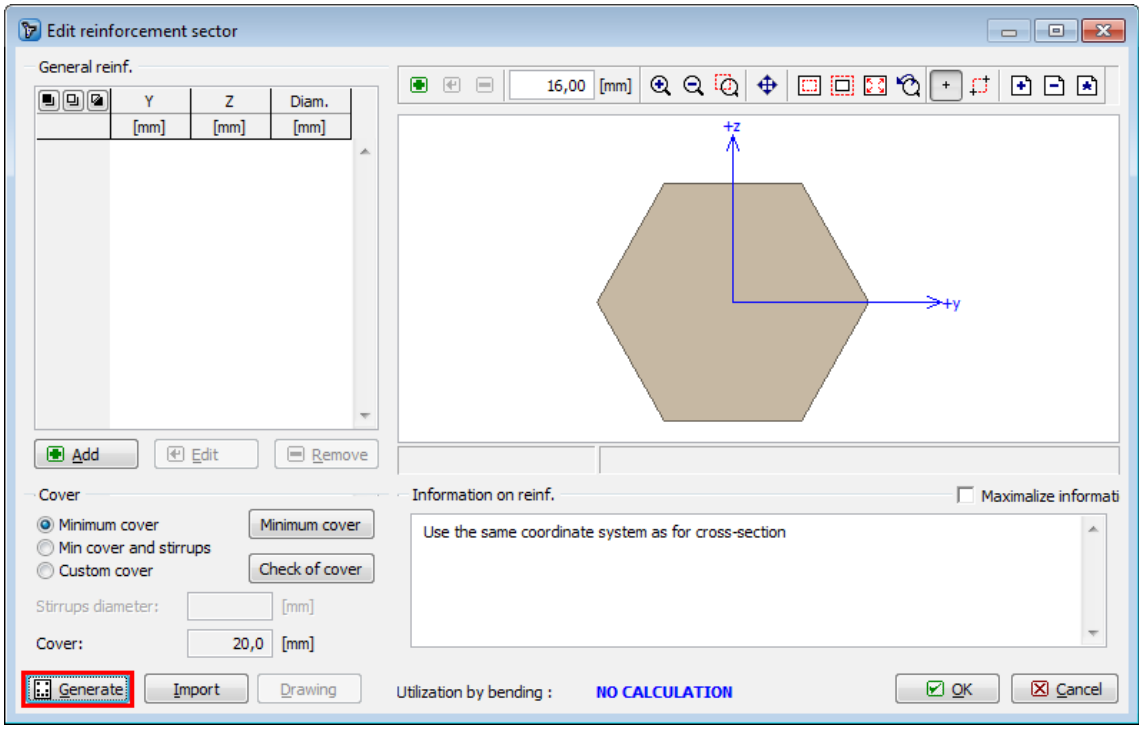
Loads

Any number of load cases for various combination types (ULS, SLS) can be defined; the program assesses each of them separately.

Reinforcement

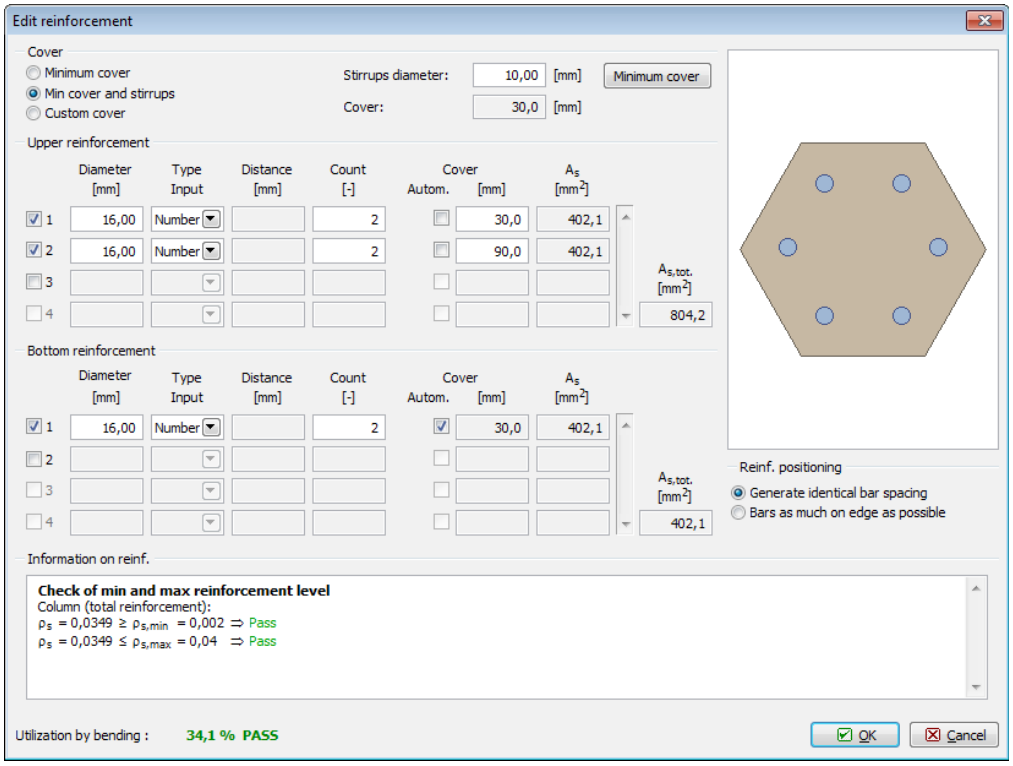
After returning to the main dialog box, we can proceed to defining the longitudinal and transversal reinforcement. We open the dialog box "**Edit reinforcement sector**" for longitudinal reinforcement definition by clicking on the "**Reinforcement**" button in the "**Section**, **Material**, **Reinforcement**" frame. In this dialog box, we can define the reinforcement either numerically in the table on the left-hand side or graphically in the right

part of the dialog box.We can also take advantage of a simplified approach by clicking " " and then the "**Generate**" button in the left lower corner of the dialog box. In our example we will use this option.



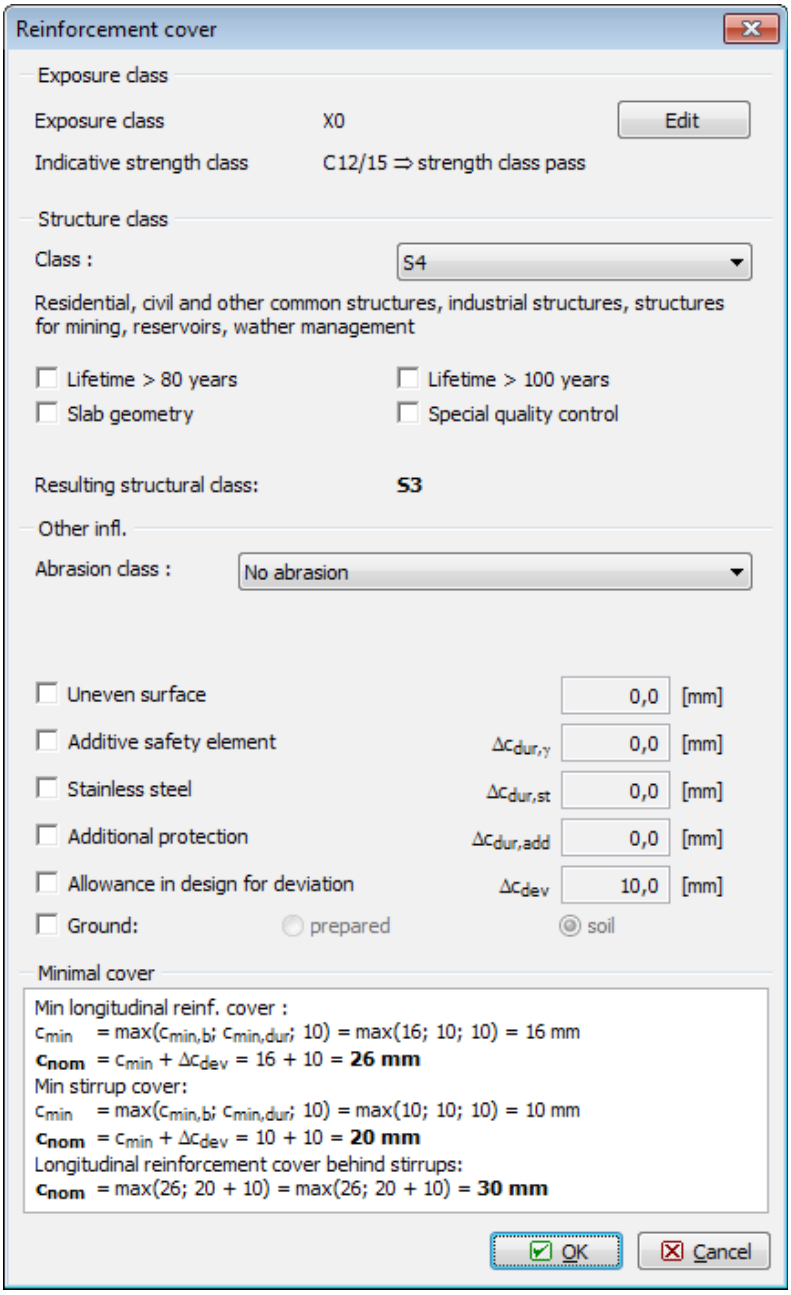
Simplified definition of reinforcement

We can now easily define required longitudinal reinforcement using *16mm* bars in each corner of the cross-section, entering 3 layers with appropriate covers:



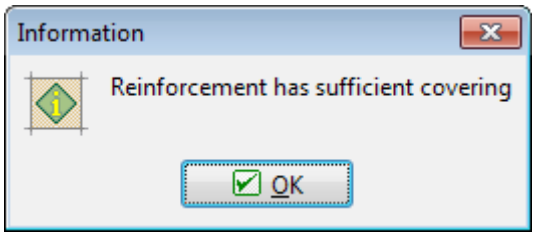
Definition of reinforcement

After defining reinforcement we can immediately check in the lower part of the dialog box that the area of the reinforcement is sufficient and passes design criteria with *34.1%* utilization by bending. In the section "**Information on reinforcement**" we can also confirm that the detailing requirements given by the code are satisfied. Finally we need to check if the covers are correctly defined. Having a column with stirrups, the option "**Min cover and stirrups**" is selected in the "**Cover**" section of the dialog box. The program will calculate the minimum required cover of the longitudinal reinforcement as sum of the minimum cover given by the code and the diameter of the stirrups. The calculation can be checked in the dialog box opened by clicking "**Minimum cover**" button:



Minimum cover calculation

As it is not necessary to change the settings of cover calculation, we can exit the dialog box by clicking "**OK**" and return to the dialog box for longitudinal reinforcement definition. To check whether the geometry of the longitudinal reinforcement satisfies requirements on minimum covers, we can run the assessment by clicking "**Check of cover**".



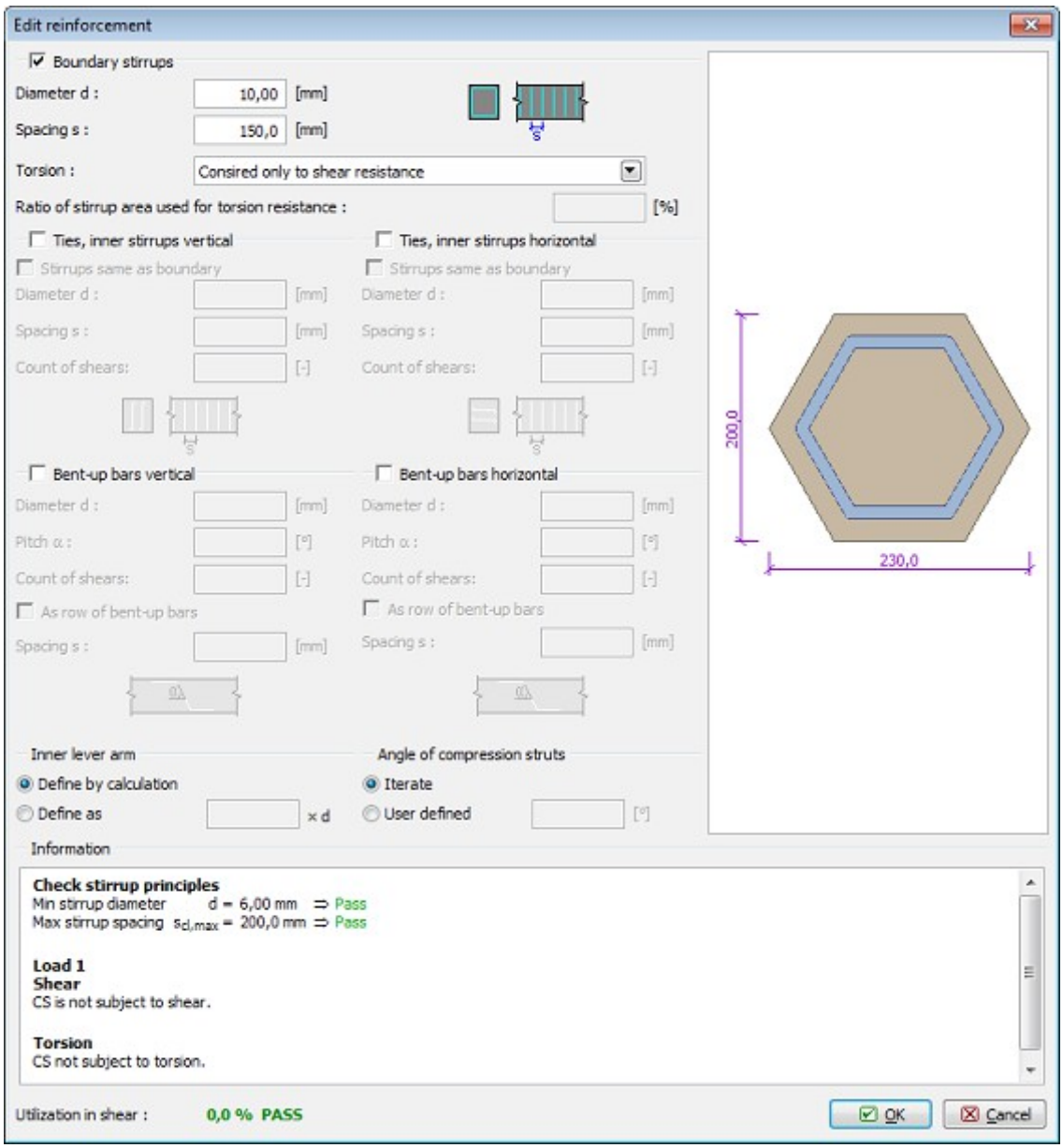
Covers check result

The check returned a positive result, therefore we can return to the main dialog box by clicking "**OK**".

Shear reinforcement

We can proceed to defining the shear reinforcement by clicking the eponymous button in the main screen.

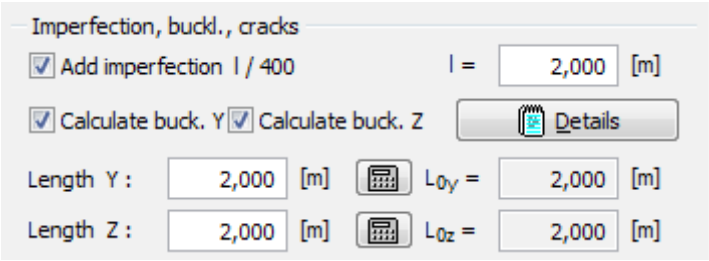
We specify *10mm* diameter boundary stirrups with *150mm* spacing:



Shear reinforcement definition

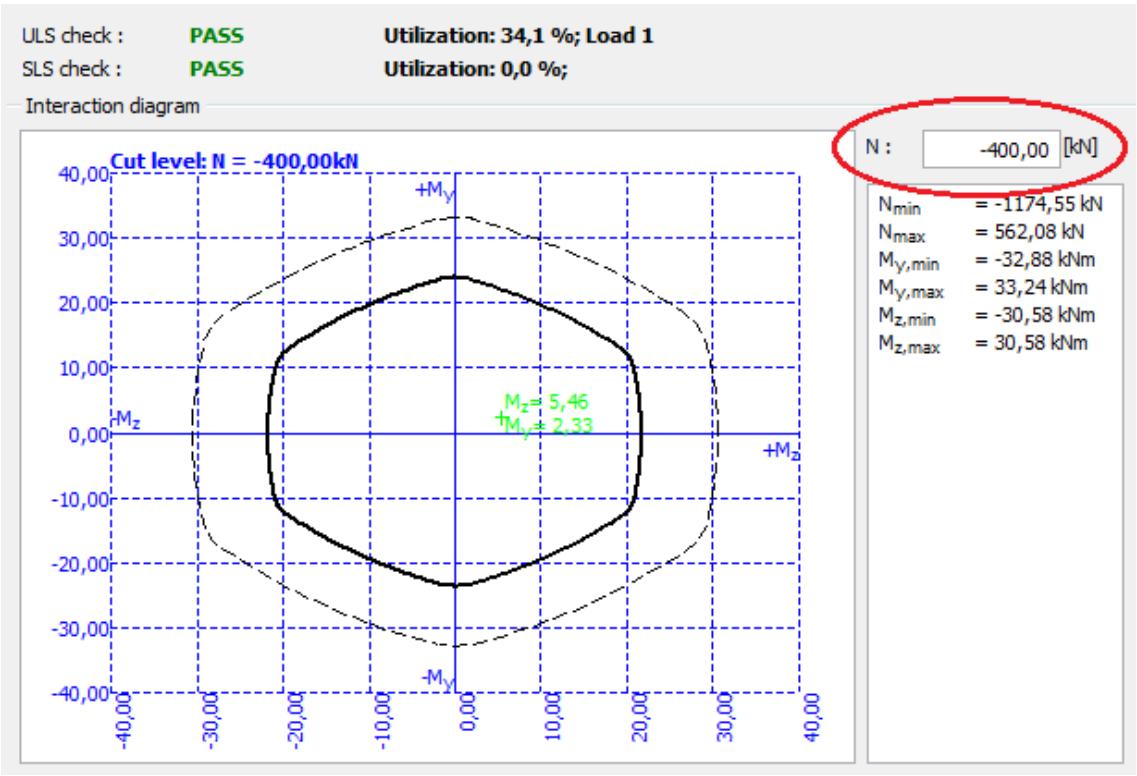
Buckling

The next step is defining the buckling parameters. Firstly we need to tick "**Calculate buck. Y/Z**" boxes followed by entering the nominal lengths of the column for both directions, based on which the effective buckling lengths will be calculated. For a column simply supported on both ends, the effective buckling lengths equal to the nominal. Defining different boundary conditions can be done by clicking the " " buttons for each direction.



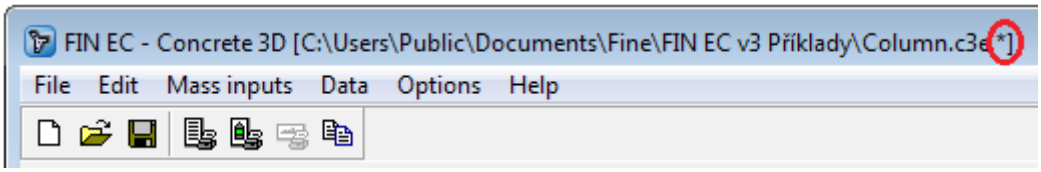
Defined buckling parameters

After defining the buckling parameters, two distinct areas of capacity are shown in the interaction diagram: thin dashed line denotes capacity of the member without influence of buckling and thick line denotes the capacity reduced by buckling effect. To check position of a defined load case in the "**Interaction diagram**" we need to enter the relevant axial force level; in our example *N= -400kN*. Program will create a cut through the interaction diagram on this level showing the position of the defined load case:



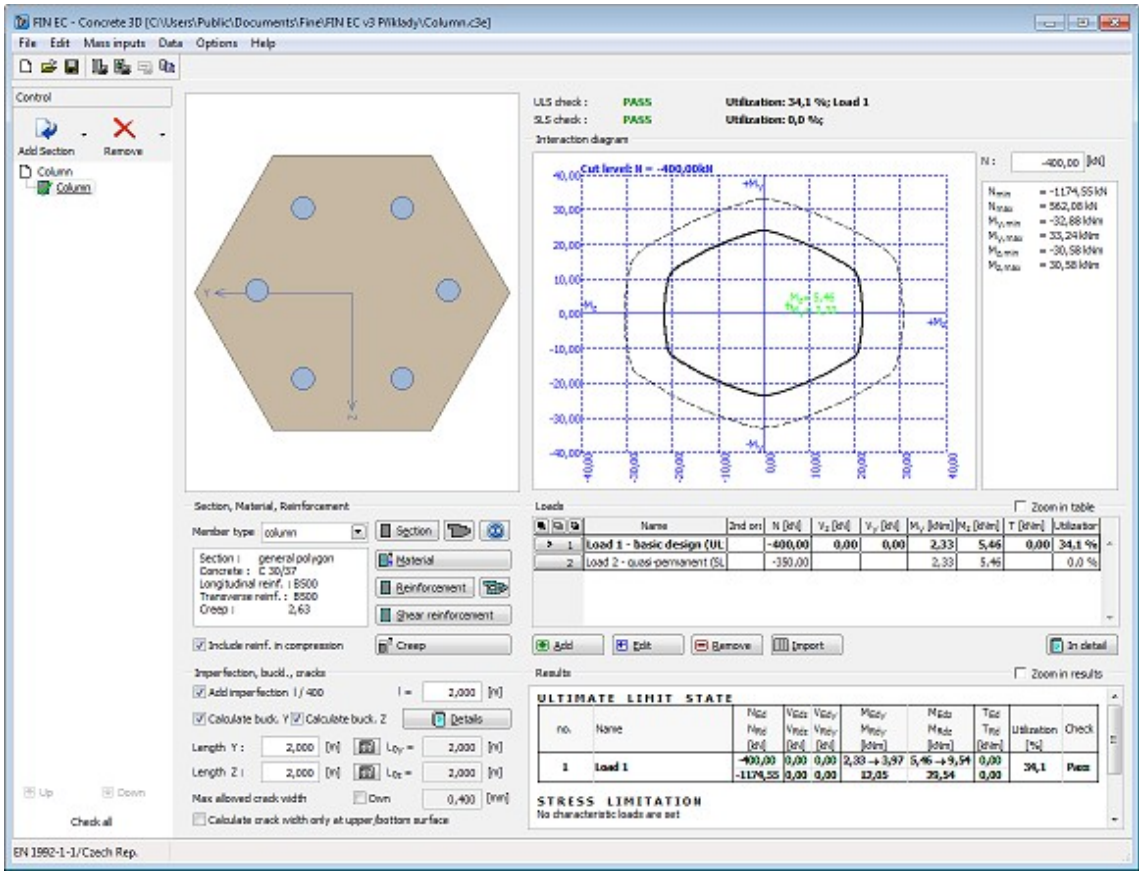
Defining a cut through the interaction diagram

Because we completed definition of all parameters, it is recommended to save the job by clicking " " on the toolbar or using shortcut "**Ctrl+S**". The actual state of the job during work may not be identical to the one saved on disk; this is indicated by "*****" in the program window header. In such case it is advisable to save the job.



Indication of non-saved job state

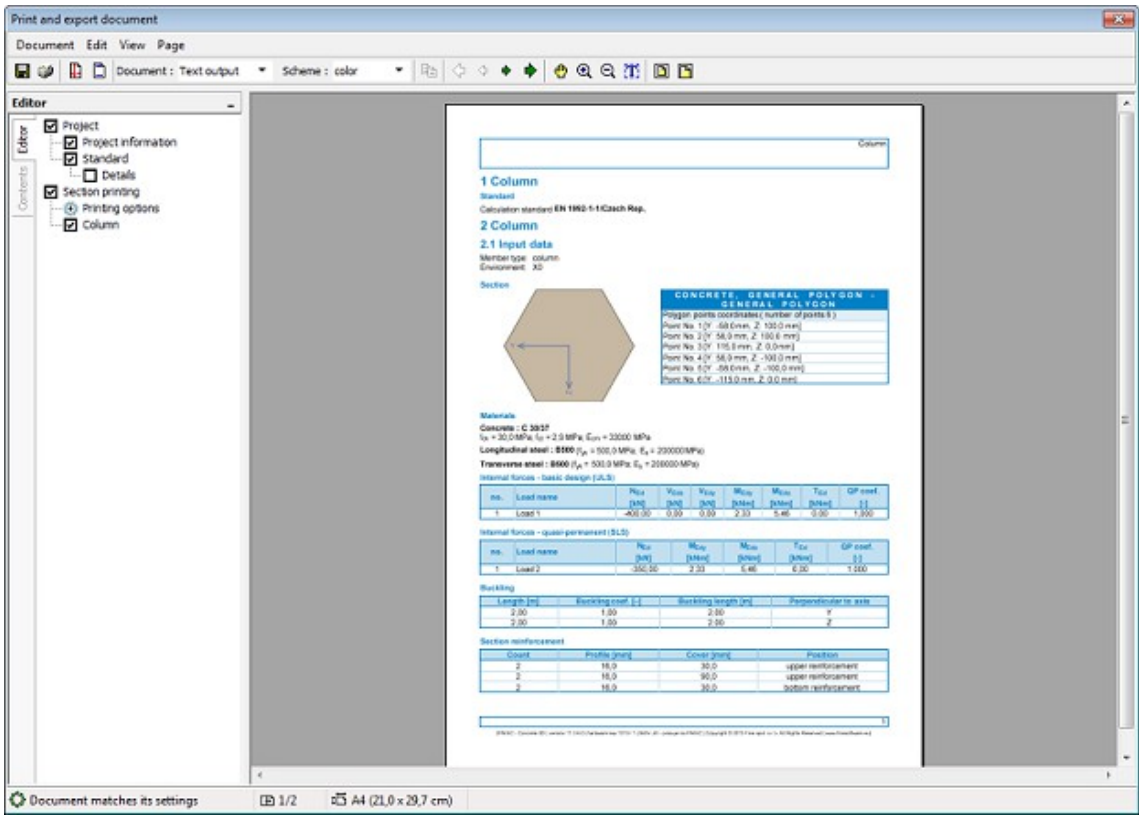
Indication of non-saved job stateAs all structural requirements have been checked and satisfied during the definition of parameters and as the main dialog box indicates that the section passes design checks in both ultimate and serviceability limit states, the job can be considered finished.



Assessed section in Concrete 3D program

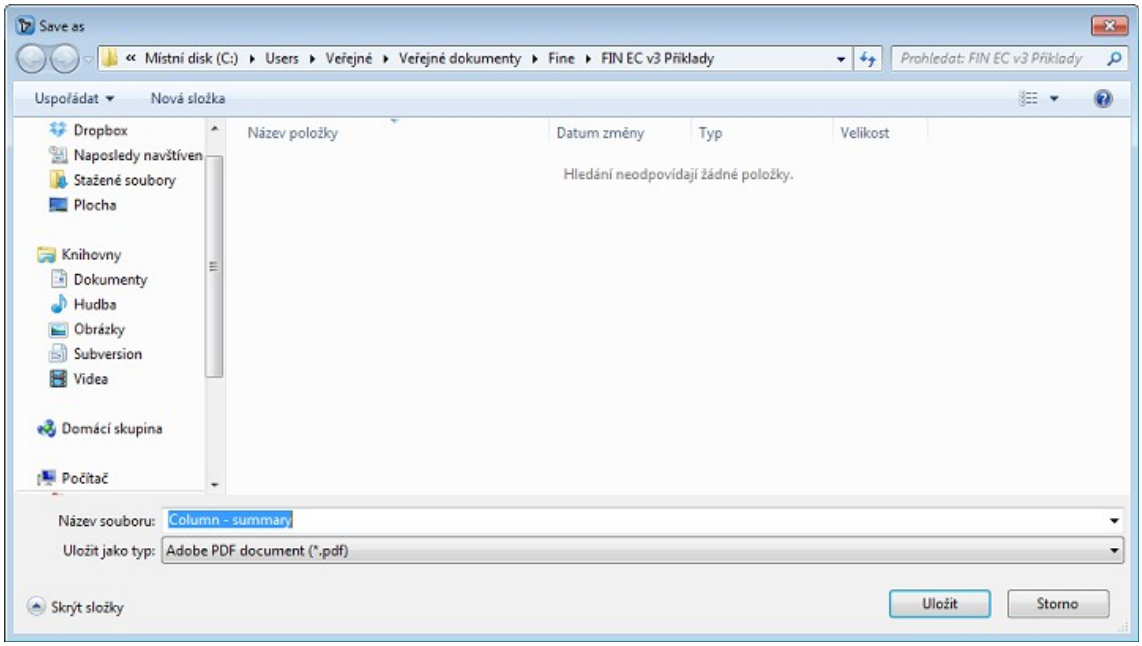
Outputs

When the job is finished and saved, we can proceed to composing the output documentation. First we print out a concise single page output which summarizes all input data and design checks results. Composition of this output is run by clicking "**File**" and "**Graphic output**" or " " in the toolbar.



Graphical output of Concrete 3D program

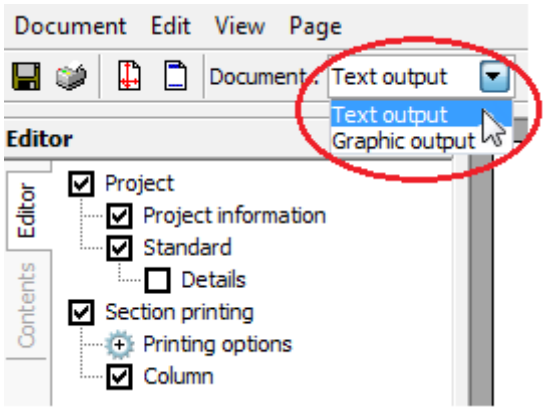
We can print the document directly by clicking " " button or save it on disk as *.pdf or *.rtf file by clicking " " button. We use the second option and save the file on disk. In the dialog box "**Save as**" we can enter the file name and select the destination folder.



*Saving file in *.pdf format*

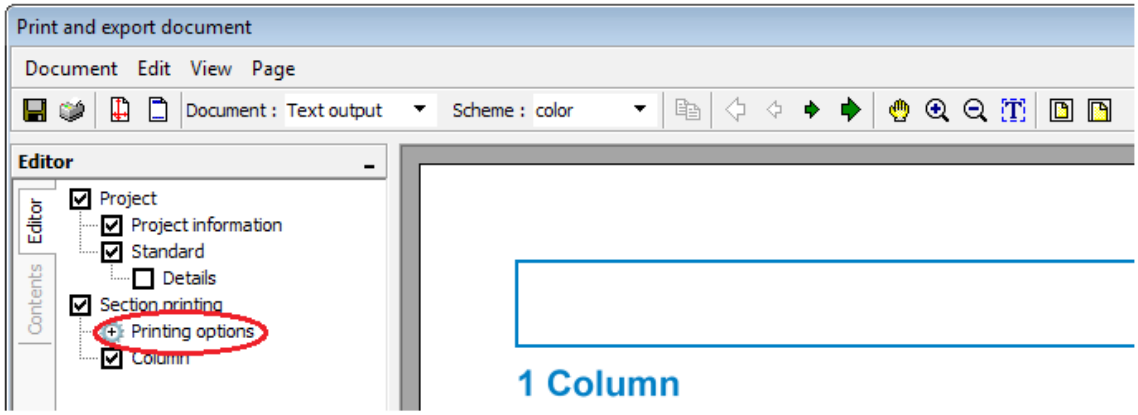
Apart from this concise document, we can also compose a detail text output by clicking the "

" button in the toolbar or selecting "**File**" and "**Text/Graphic output**" in the main menu. However, as we are still in the print and export document dialog box we can change the document type to text output directly in the toolbar's "**Document**" drop-down list.



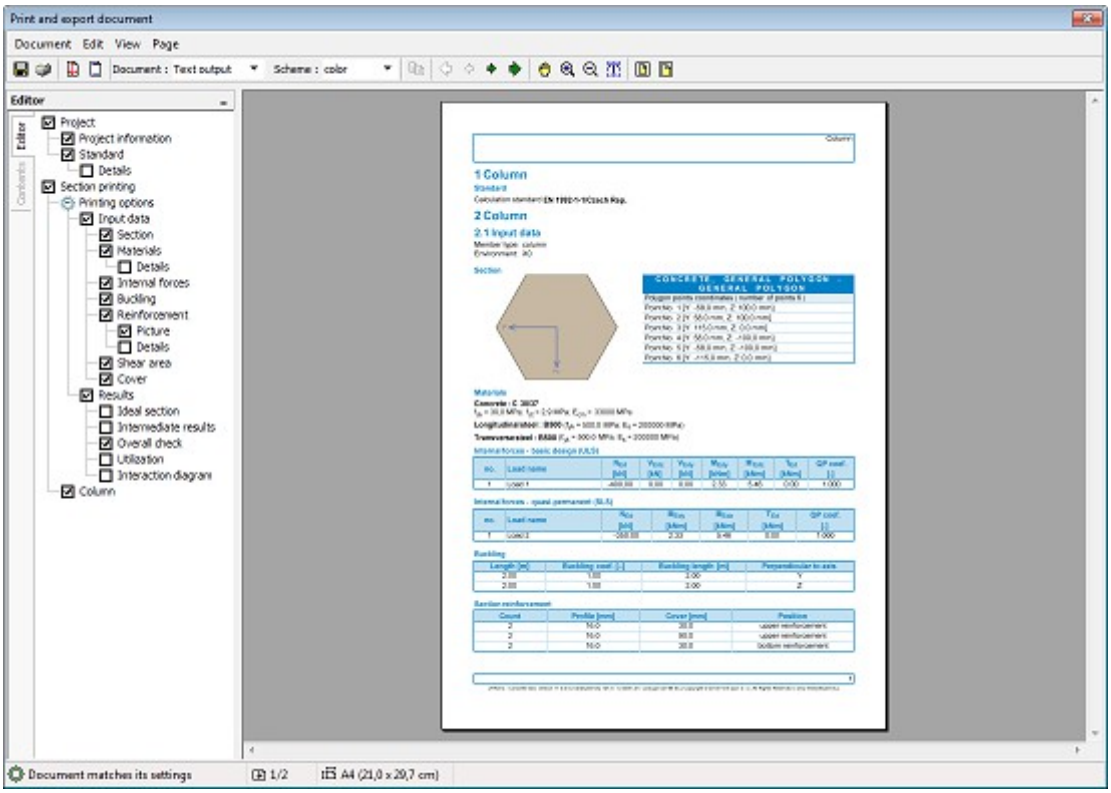
Change of output type

After switching to the "**Text output**" mode we can set in the "**Editor**" which parts of the assessment will be included in the output and how detailed the output shall be.



Printing options of text outputs

Program will immediately re-generate the output to reflect each change made in the settings in the tree on the left-hand side. Once the output contains all required information, we can again save the document on disk.



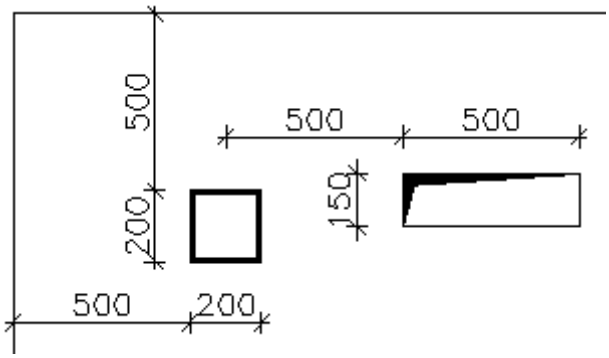
Generated text output

Completing the outputs generation, our work is done.

Punching shear

Task

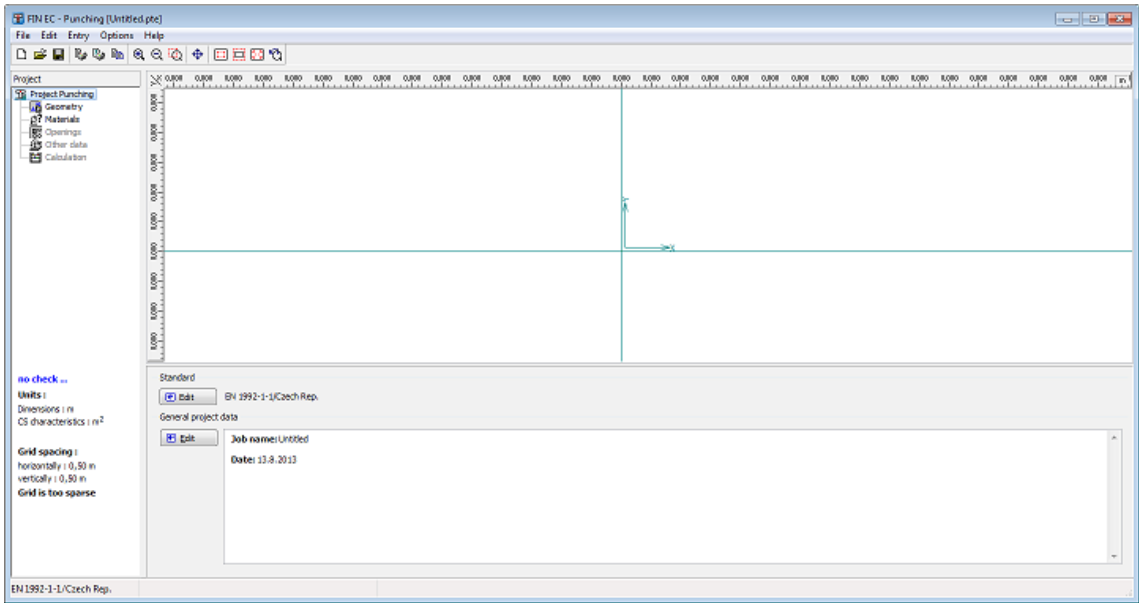
In this example, the task is to design reinforcement against punching of a corner column of 200 x 200 mm square cross-section through a 200 mm thick reinforced concrete slab. The column is located 500 mm from the edges of the slab, which is weakened by an opening of 150 x 500 mm located as per the following sketch. Concrete strength class C25/30 is used in the design.



Problem's geometry

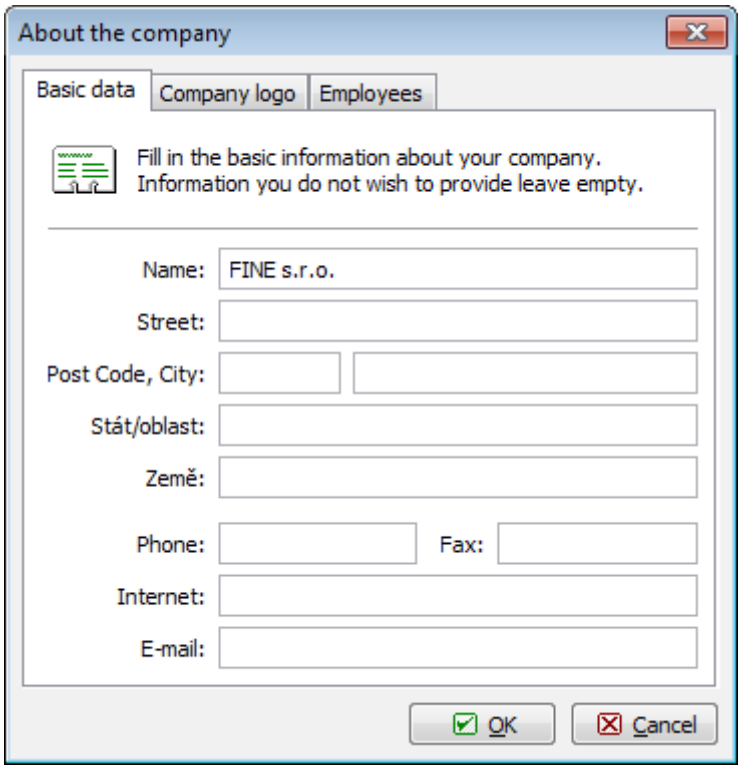
Project data

In the main program window, all essential project data can be entered in the frame in the bottom part. Definition modes can be switched between using the tree on the left hand side; the rest of the window serves as a model space.



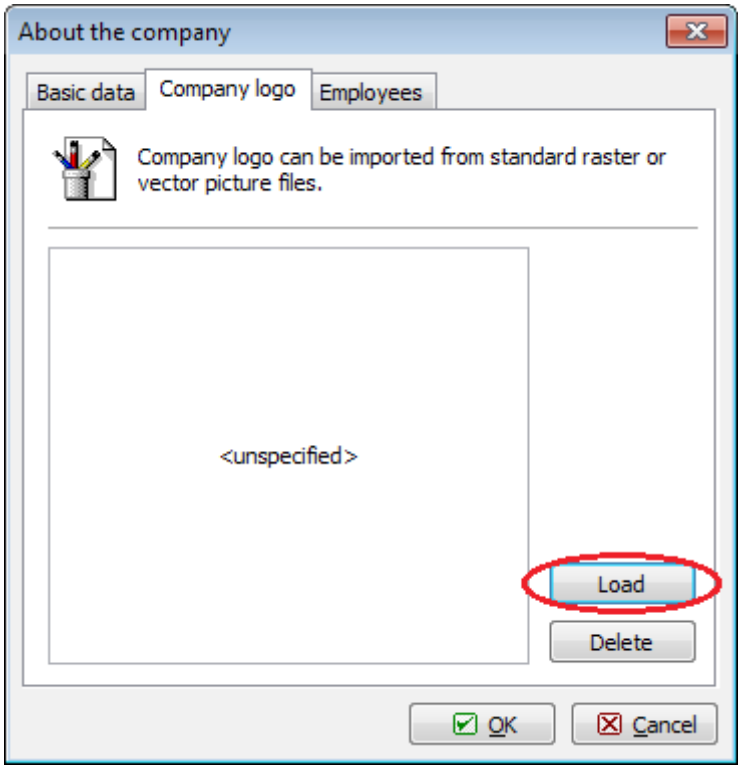
Main window of the program "Punching"

First we enter the company data which will be later used when composing the output documentation. The dialog box **"About the company**" can be run from the main menu selecting "**Settings**". In the first tab for instance, we can enter company's name and address.



Essential company data in the "About the company" dialog box

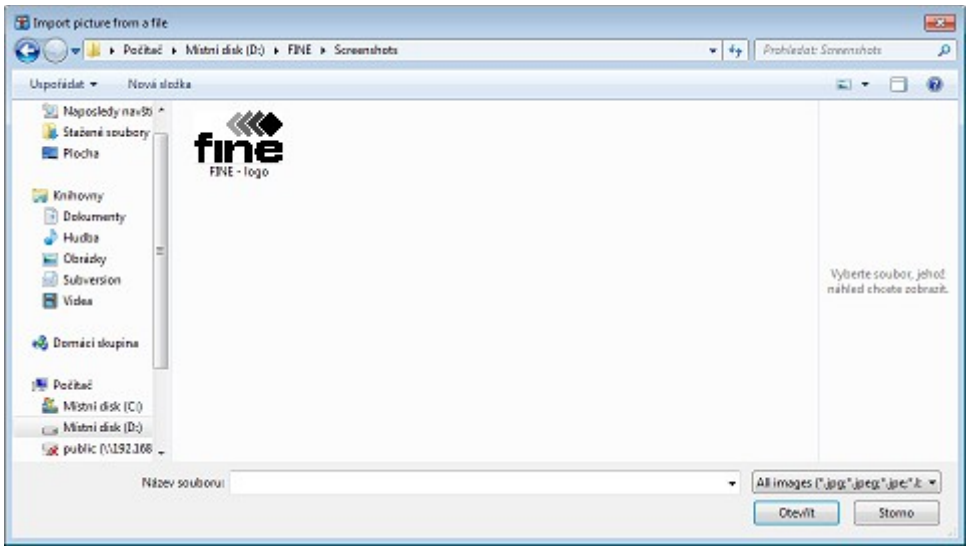
In the second tab we can enter the company's logo which can later be displayed in the header of the output documentation. The logo can be loaded as an image of any usual format such as *.bmp, *.jpg, *.ico etc. To import the logo we run the dialog box by clicking "**Load**".



Button for loading the logo file

For selecting files, the standard Windows user interface is used. After opening the folder

containing the file, we load the file by clicking the "**Open**" button.

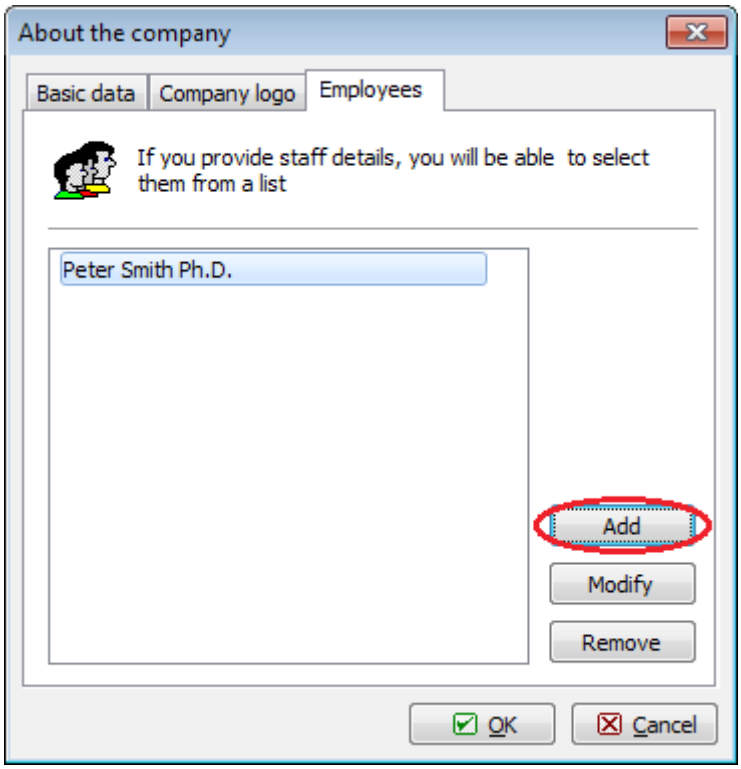


Logo file selection

A preview of the logo is subsequently displayed; its size will be automatically adjusted to the graphic layout of the program's output.

A logo loaded to the program

In the last tab "**Employees**", we can enter a list of company's employees from which we can select the author of the project. New employees are entered by clicking the "**Add**" button.

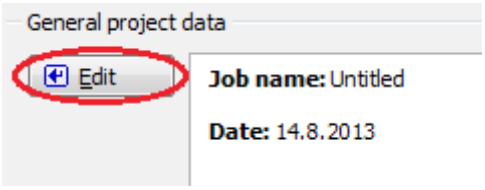


Entering a list of employees

We exit the dialog box "**About the company**" by clicking "**OK**". The company's data are shared with all EC programs and can be therefore used in other programs of the package.

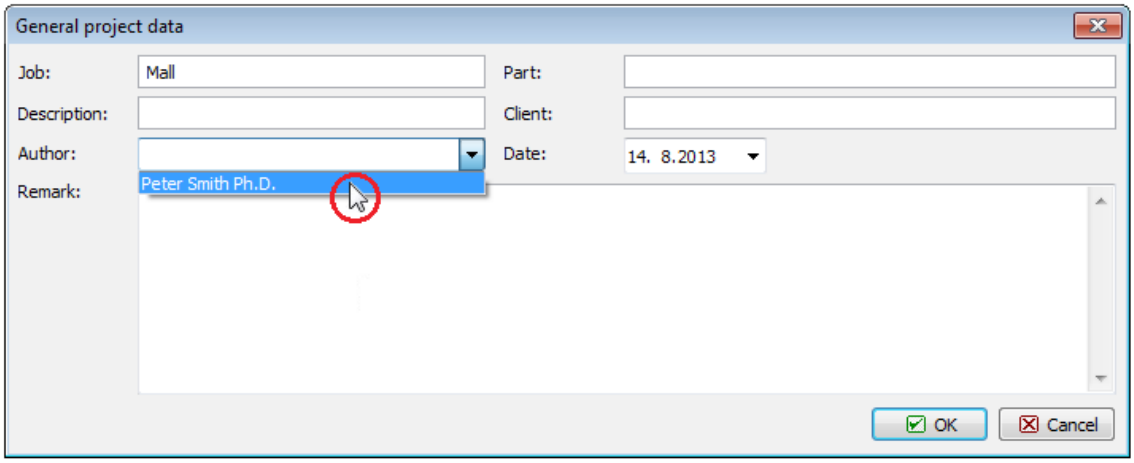
Basic settings

After entering data in the "**About the company**" dialog box we return to the main screen. Here we can enter essential information about the job in the "**General project data**" dialog box which opens by clicking the "**Edit**" button.



Opening "General project data" dialog box from the main window

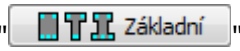
In the "**General project data**" dialog box we can enter e.g. job's name, description, remarks etc. It is also possible to select the author of the project from the list of the employees which we created in the "**About the company**" dialog box. All this information can be later displayed in the headers or footers of the output documents.



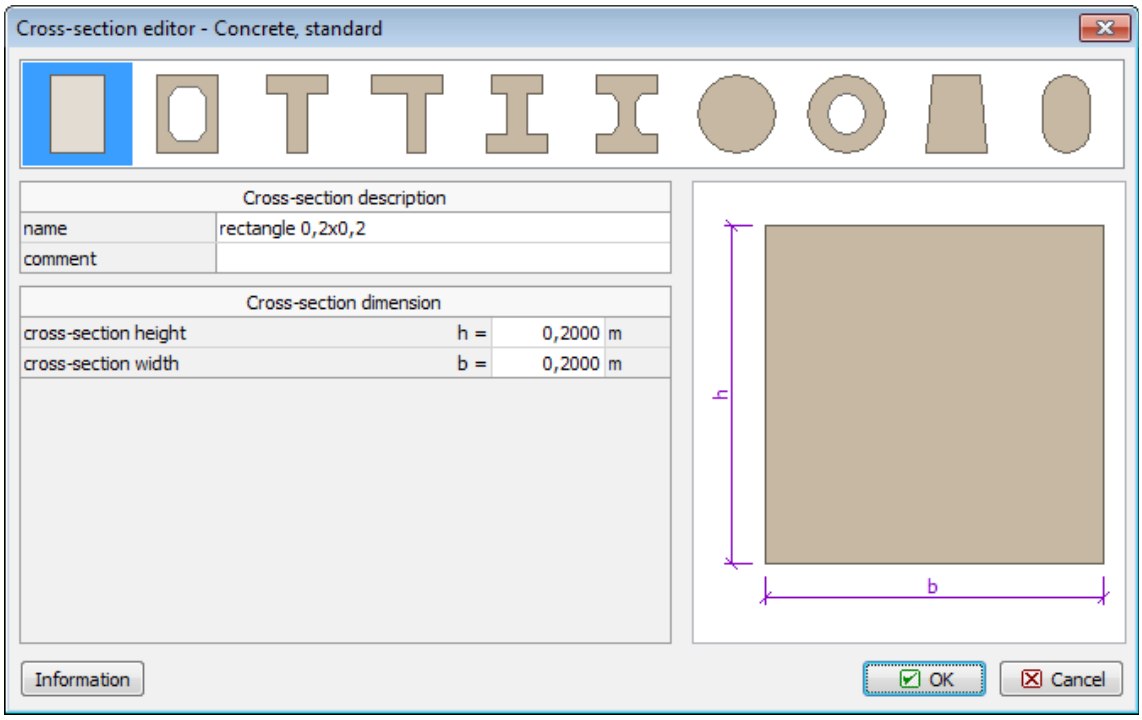
Selecting job's author from list in "General project data" dialog box

Once we have finished entering the project data, we exit the dialog box by clicking "**OK**". Then we can proceed to the next step by clicking the "**Geometry**" button in the tree on the left-hand side.

Geometry

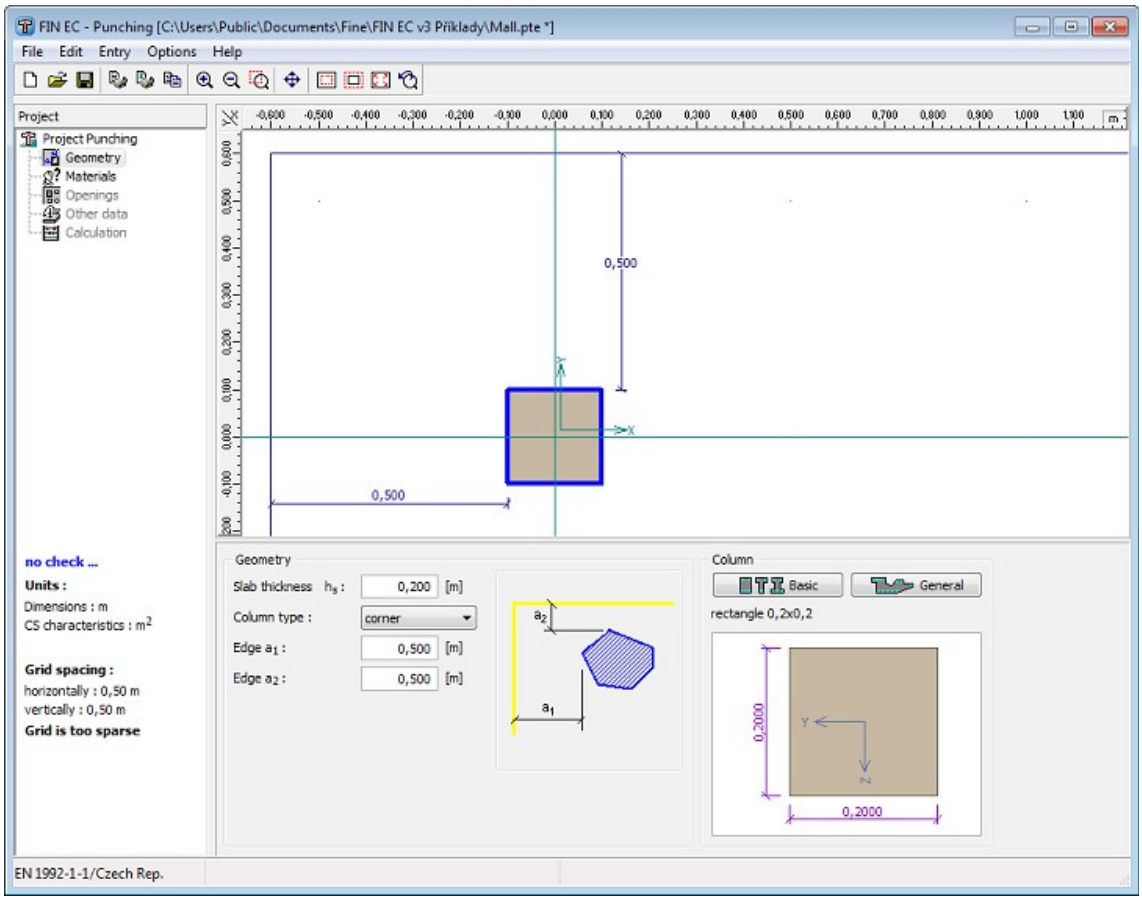
In this dialog box, we first select the column's cross-section. Clicking the " 

" button we open the catalogue of pre-defined shapes. Then we select a rectangle and enter the dimensions.



Catalogue of pre-defined cross-section shapes

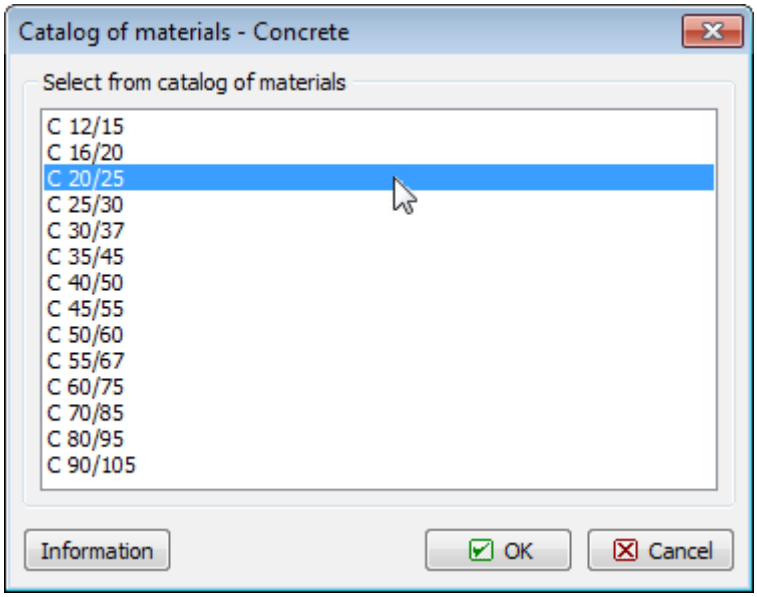
In the "**Geometry**" section of the main screen we enter the slab thickness, select the "**corner**" column type and define the edge distances. The structural layout instantly appears in the model space.



Defining job's geometry

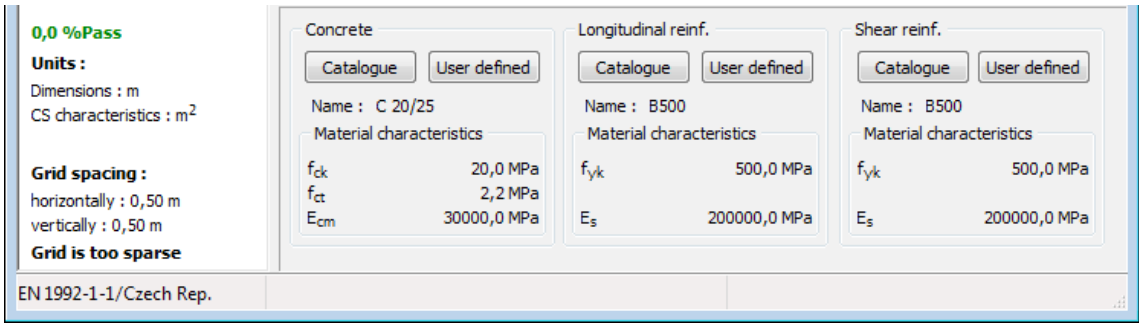
Materials

Proceeding to the next item of the tree - "**Materials**", another sub-window appears in the main window's bottom part. In this section we need to specify materials for concrete and slab's longitudinal and punching shear reinforcement. As we are using a standard concrete class, we click on the "**Catalogue**" button and open the list of available concrete strength classes. We select C20/25 and exit the dialog box by clicking "**OK**".



Concrete strength class selection

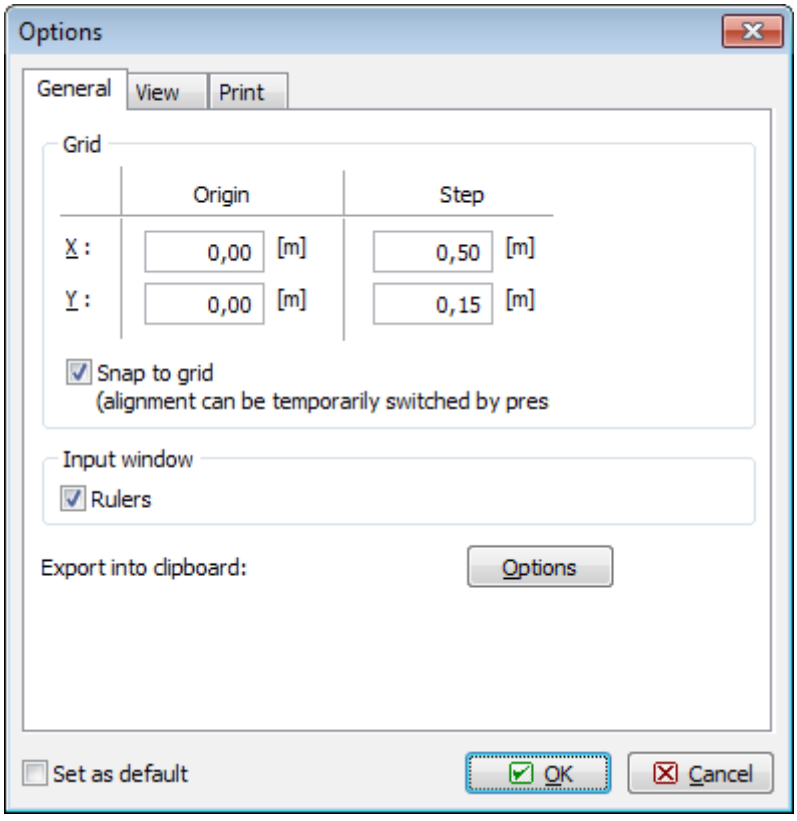
We proceed analogically to specifying reinforcement material. For both longitudinal and shear reinforcement we will select the "**B500**" grade.



Materials' definition sub-window

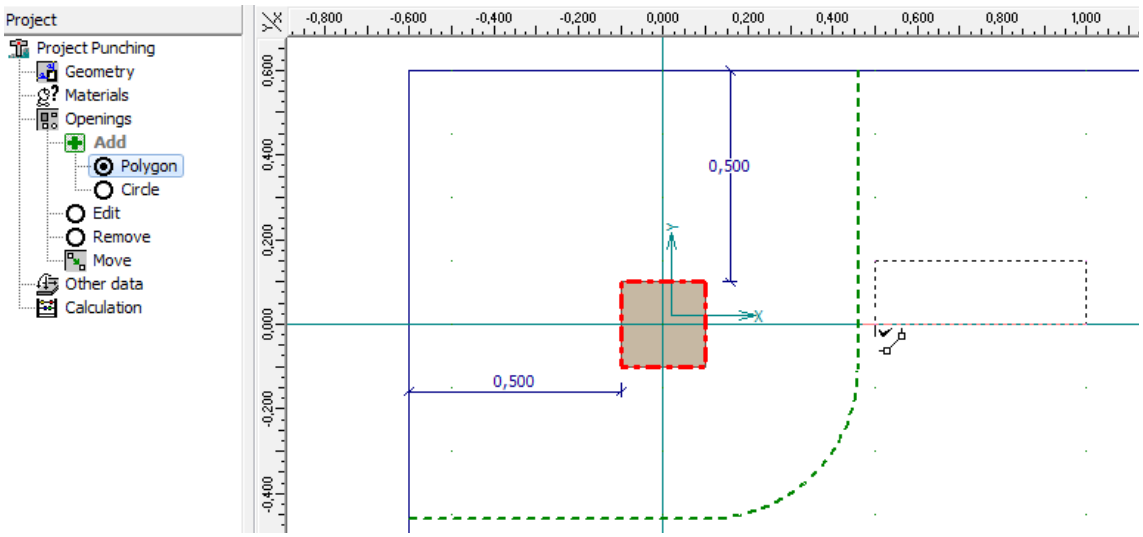
Openings

To simplify definition of the opening we first open the "Options" dialog box by selecting "**Settings**" in the main menu. In the tab "**General**" we adjust the grid to suit our needs by defining the step as 0.5 m in the X direction and 0.15 m in the Y direction. We exit the dialog box by clicking "**OK**".



Adjusting grid step in the "Options" dialog box

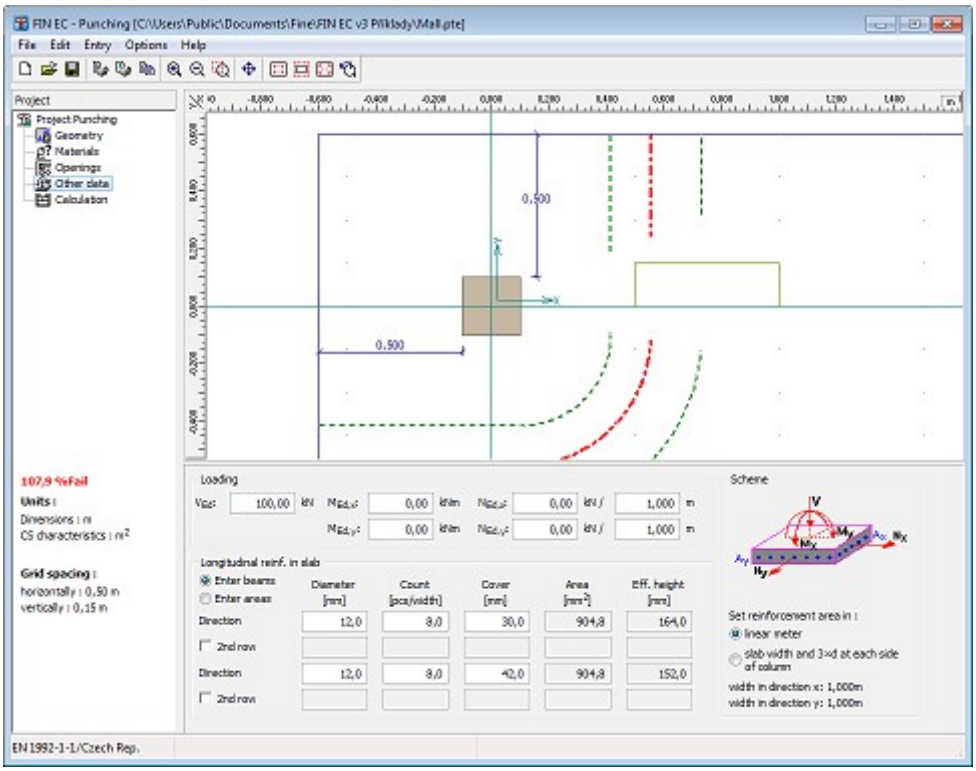
Now we can switch in the tree to "**Openings**" and "**Polygon**" and draw the contour of the opening directly in the model space clicking in each of its four corners. To finalize the contour we click again on the first corner.



Definition of opening in model space

Other data

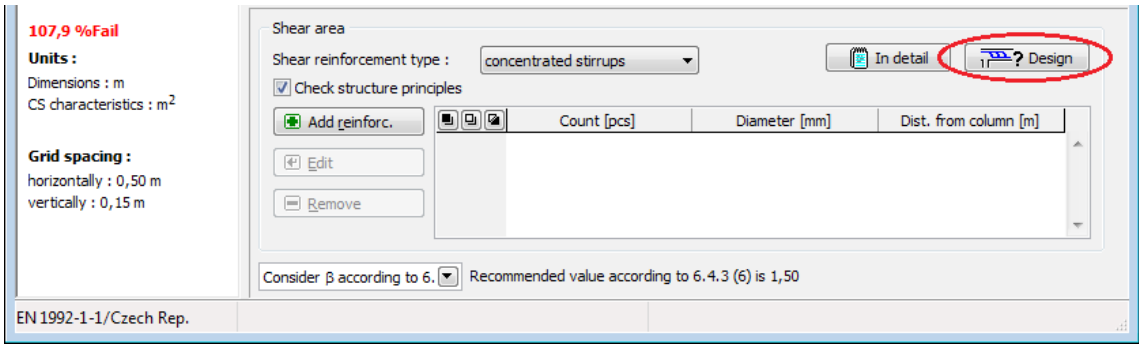
In this section of the tree we define the loading and longitudinal reinforcement of the slab as V_{Ed} = 100 kN and 8 of 12 mm diameter bars per meter run.



Slab loading and reinforcement definition

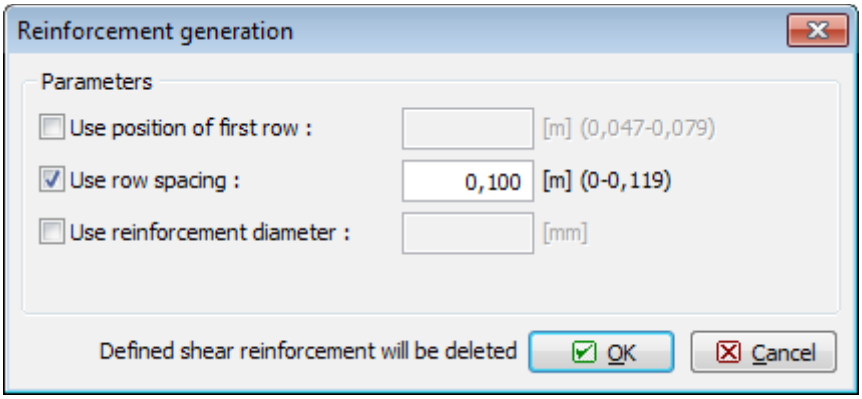
Calculation

Switching to the last item of the control tree "**Calculation**" we can proceed to carrying out the design checks of the punching shear reinforcement. First we need to select the shear reinforcement type; in our example we will use "**concentrated stirrups**". Then we can either define the reinforcement manually in the table in the definition frame or we can let the program design the reinforcement automatically. Choosing the latter, we run the automatic reinforcement design by clicking the "**Design**" button.



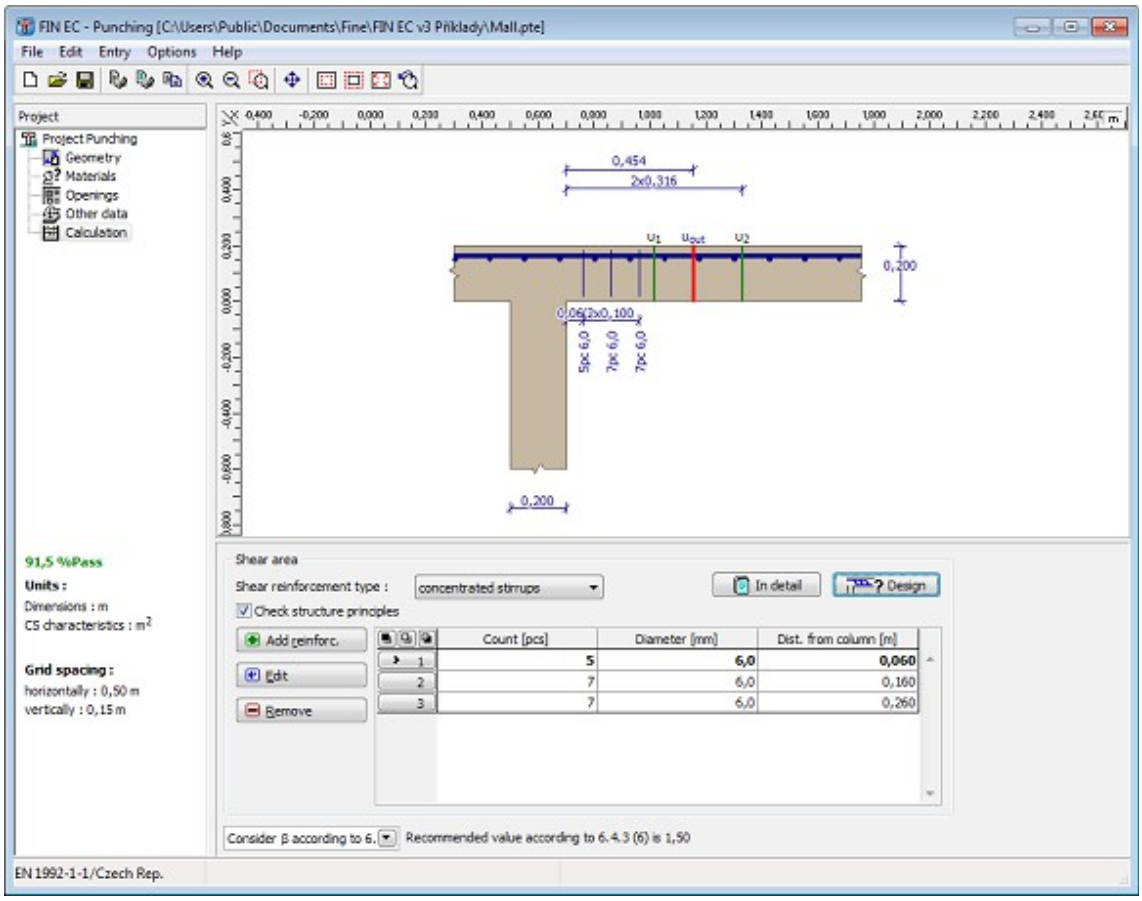
Button for automatic design of punching reinforcement

In the "**Reinforcement generation**" dialog box we can define invariable reinforcement parameters; other parameters will be defined automatically. Ticking the "**Use row spacing**" box and entering a value we define fixed stirrup spacing 100 mm; the position of the first row and the stirrups diameters we leave to be designed automatically.



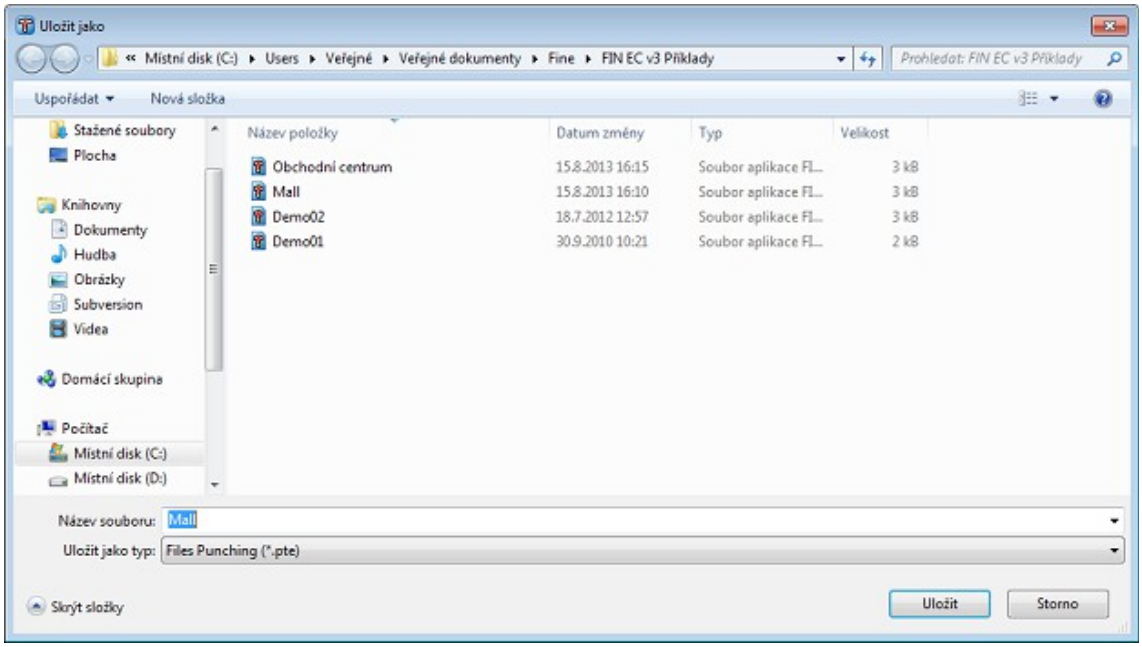
Dialog box for automatic design of reinforcement

After closing the dialog box by clicking "**OK**" the reinforcement is designed to reach required capacity and fulfil all detailing requirements.



Designed and checked punching shear reinforcement

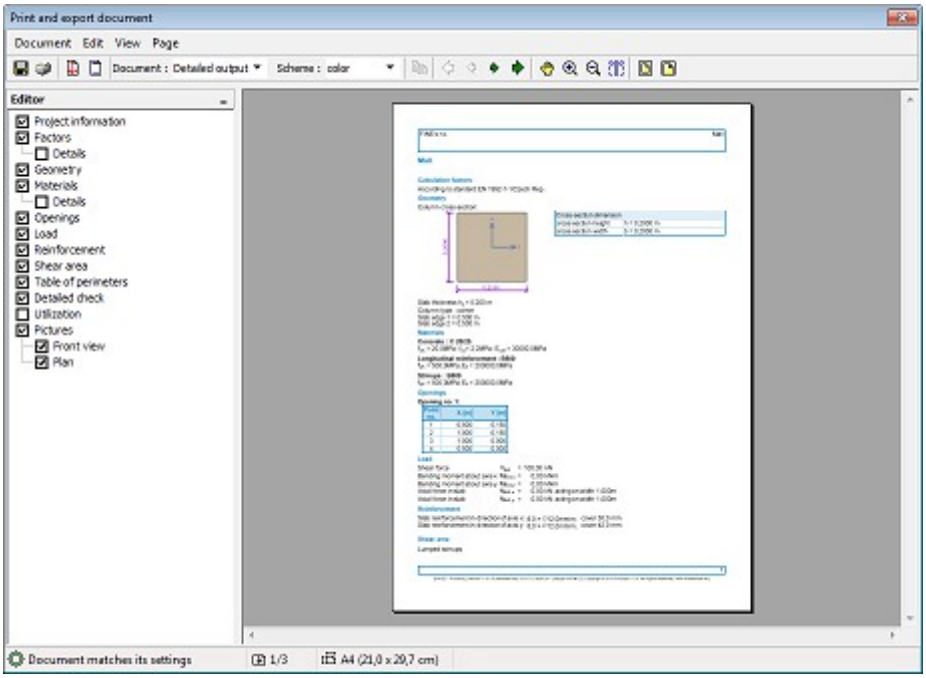
Finally, we save the job using e.g. the "**Ctrl+S**" shortcut. The program runs the standard "**Save as**" dialog box where we find and select the destination folder, enter the name and save the job.



"Save as" dialog box.

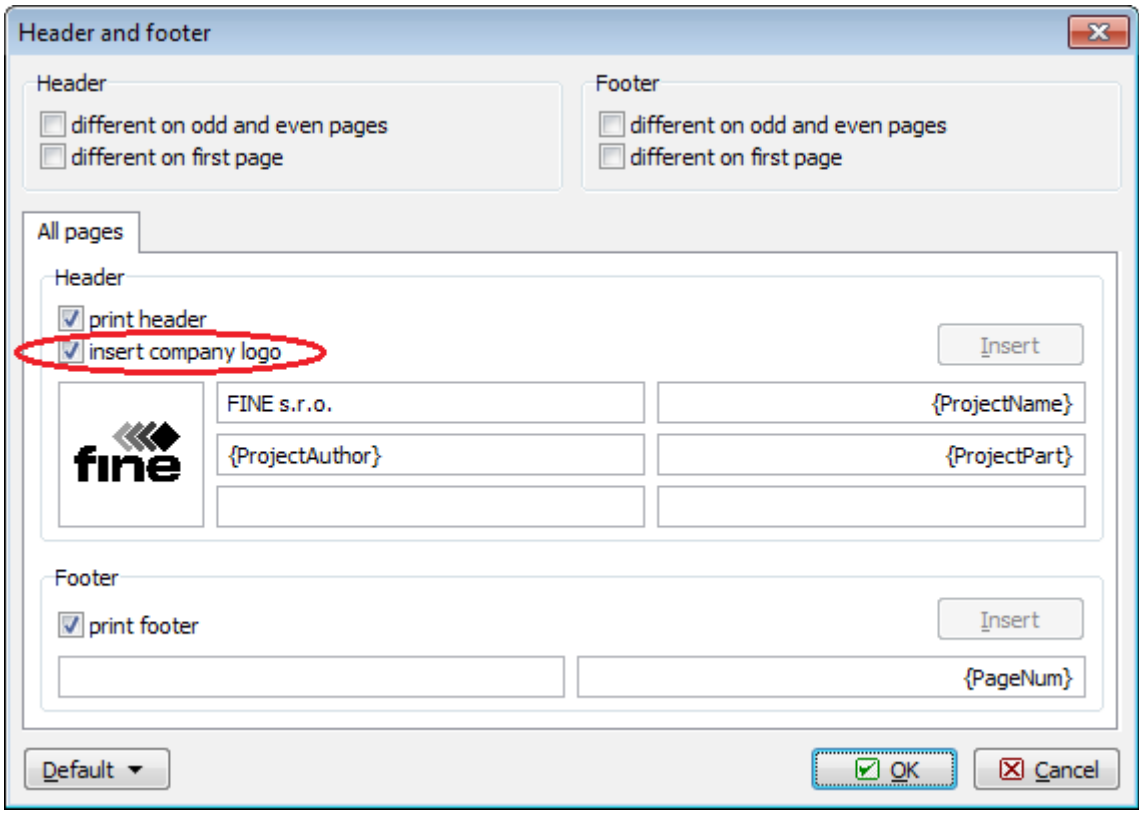
Outputs

Composition of the output documentation follows the same procedure as for the other FIN EC programs. We can compose both graphical and/or text output; for the text output we can select in the control tree which chapters of the documentation are to be printed out. We can print the composed document directly by clicking the " " button or save it as a *.pdf or a *.rtf file by clicking the " " button.



Output documentation composition window

At the beginning we imported the company's logo into the program, now we can switch on displaying the logo in the documents header in the "**Header and footer**" dialog box. We run this dialog box by selecting it in the "**Document**" section of the main menu or directly by clicking the " " button in the toolbar.



Displaying logo in document's header

After ticking the "**insert company logo**" box, the logo appears in the headers of all documents.

Header with the company's logo

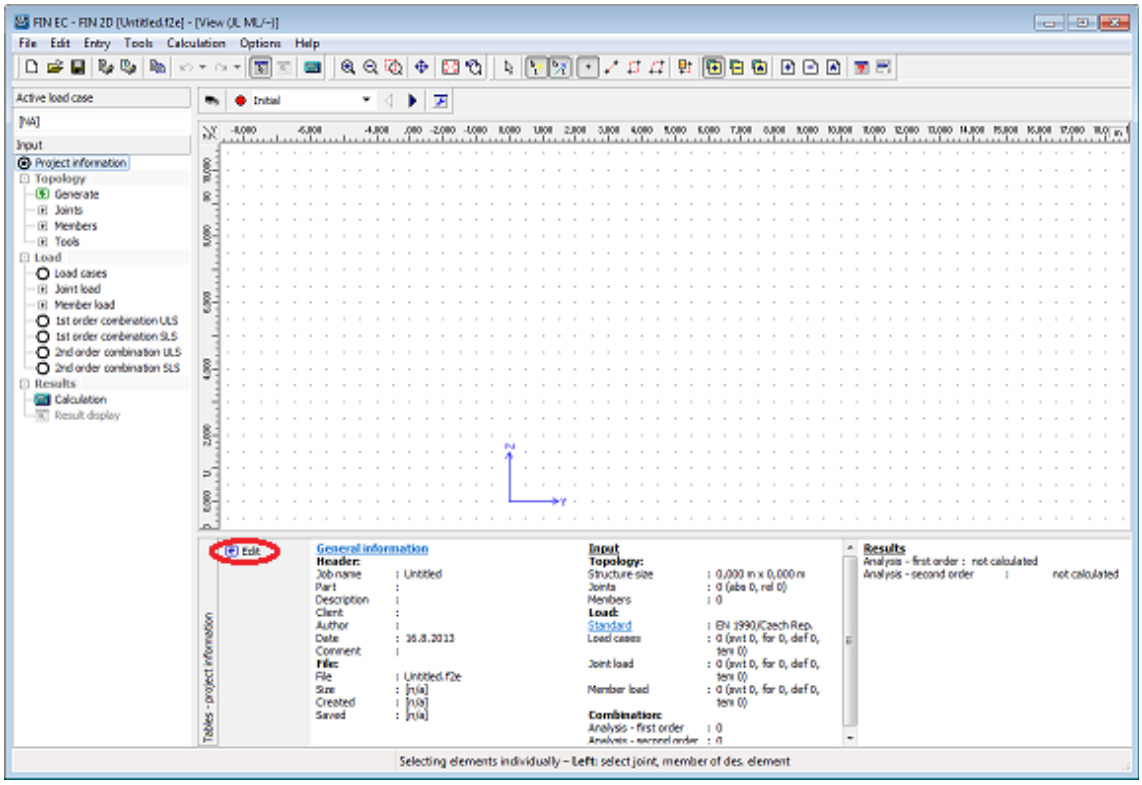
Timber truss

Task

In this example, the task is to design a symmetric roof timber truss of *13 m* length and *25 degrees* of the roof pitch. The truss consists of timber members of class *C24* and *40 mm* thickness; the spacing of the trusses is *1 m* centres. The truss is subject to dead load *0.2 kN/m* from both roofing and ceilings and to snow load *1.0 kN/m* according to the Snow area 2 of the Czech snow map.

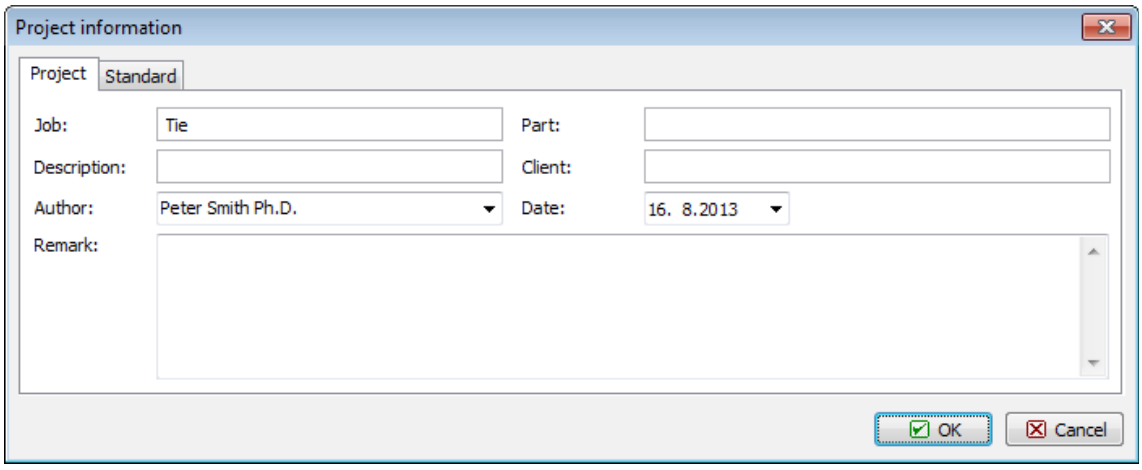
Setting up project

After running the program 2D, the main screen appears, consisting of the model space on the right-hand side, the control tree on the left-hand side and the input table in the bottom part. The table displays the project information which can be later used in headers and footers of the output documents. To enter or edit the project information, we can run the relevant dialog box by clicking the "**Edit**" button.



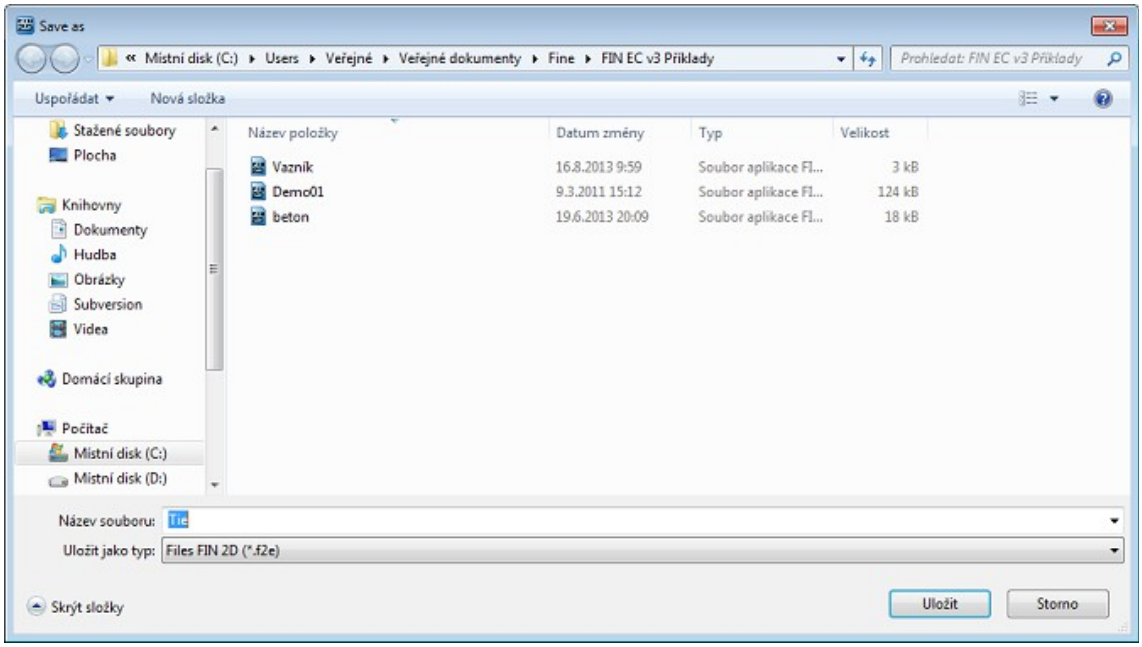
Button for running the "Project information" dialog box

In the dialog box we can enter e.g. the job title or the project author. After entering all necessary data we exit the dialog box by clicking the "**OK**" button.



"Project information" dialog box.

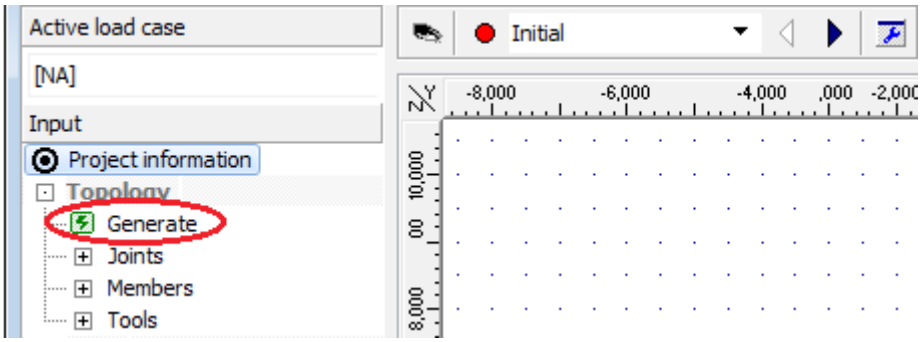
Before proceeding with work we should save the job using e.g. the "**Ctrl+S**" shortcut. The file name is entered and the destination folder selected in the standard 'Save as' window.



"Save as" window.

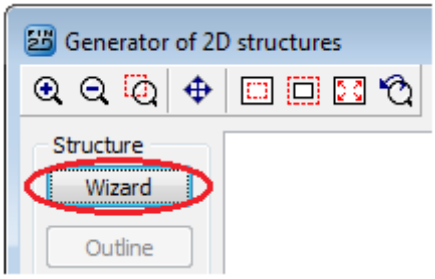
Structure generation

We can define the truss geometry either by entering individual nodes and members or we can simply make use of the Generator of 2D structures. In our example we will choose the latter approach. The generator is run by clicking the "**Generate**" button in the "**Topology**" section of the control tree.



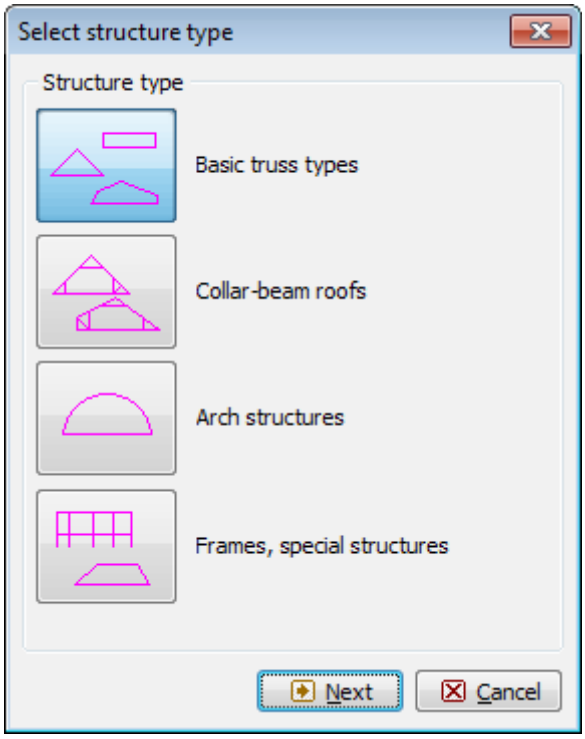
Running "Generator of 2d structures"

To generate the structure in the generator, we first click the "**Wizard**" button in the top left corner of the dialog box.



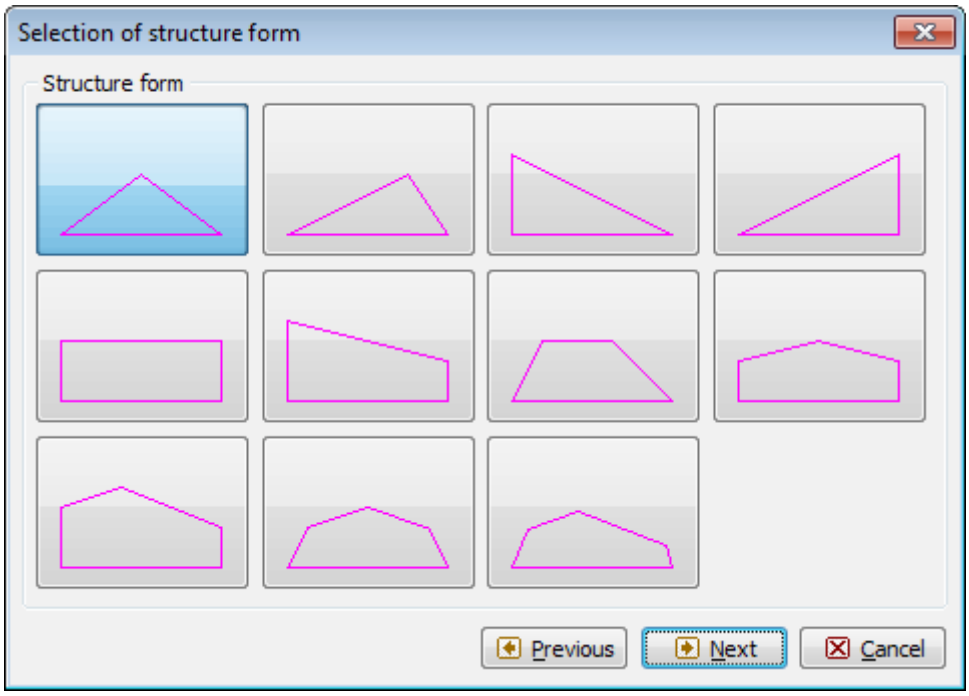
Running wizard in Generator of 2D structures

A dialog box appears enabling us to select one of the basic types of the structure. We select Basic truss types and proceed to the next step by clicking the "**Next**" button.



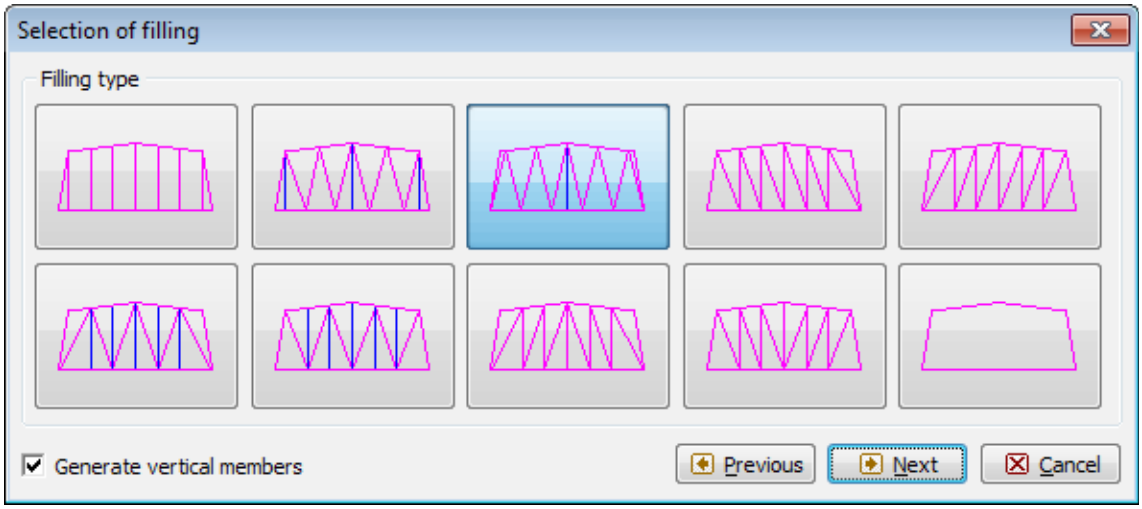
Selecting structure type

In the following dialog box, we select the desired form of the truss and click the "**Next**" button.



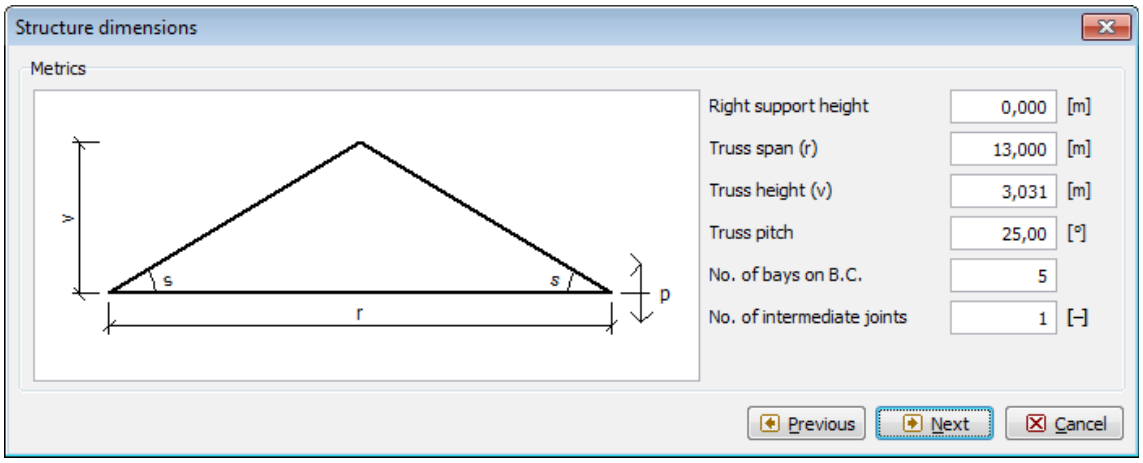
Selecting structure form

The following dialog box offers a selection of basic types of filling members' layouts. We select the desired filling type and in the bottom left corner we switch off automatic entering of verticals by unticking the "**Generate vertical members**" box.



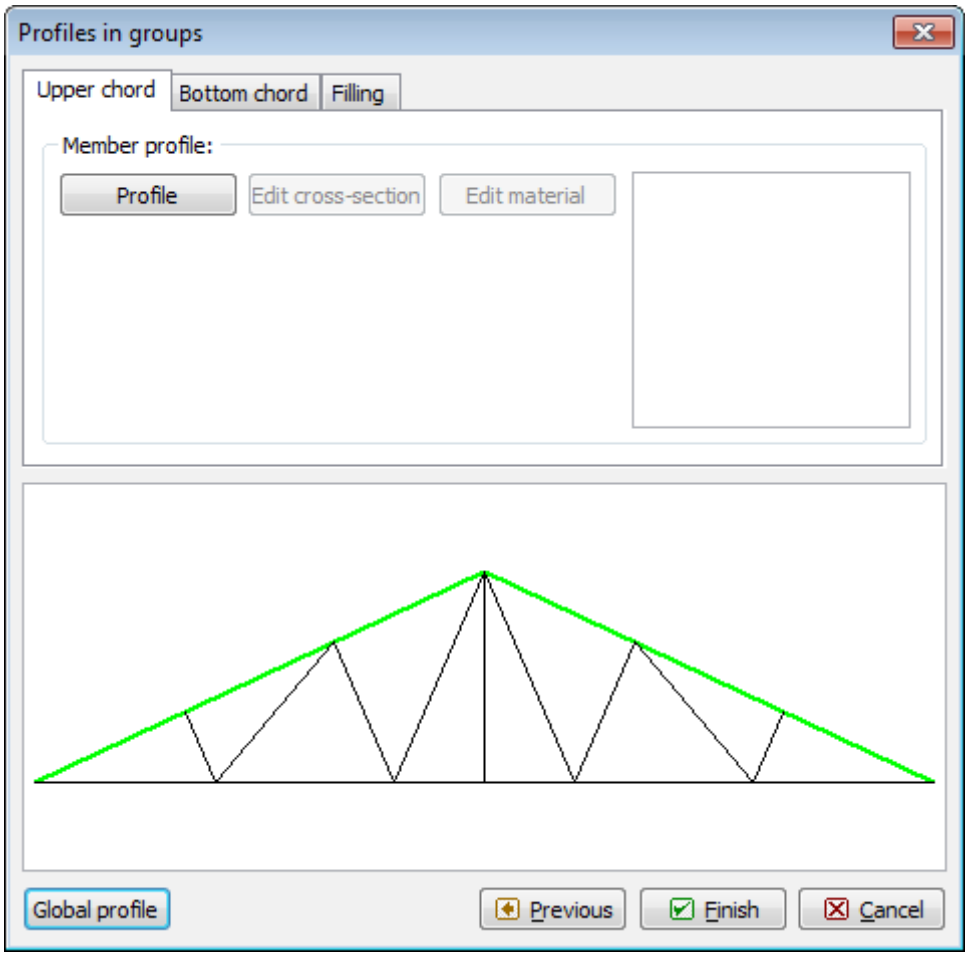
Selecting filling type

The next dialog box "**Structure dimensions**" enables defining the main dimensions of the truss. If we enter the pitch and the length, the program will automatically calculate the height of the truss. In the field "**No. of bays on B.C.**" we define into how many segments the bottom chord will be divided by nodes. Additional nodes may be inserted between the main nodes of the upper and the bottom chords. In these nodes, the exact values of internal forces and deformations will be calculated and additional nodal forces can be defined. In the field "**No. of intermediate joints**" we can define how many nodes will be added to each segment.



"Structure dimensions" dialog box.

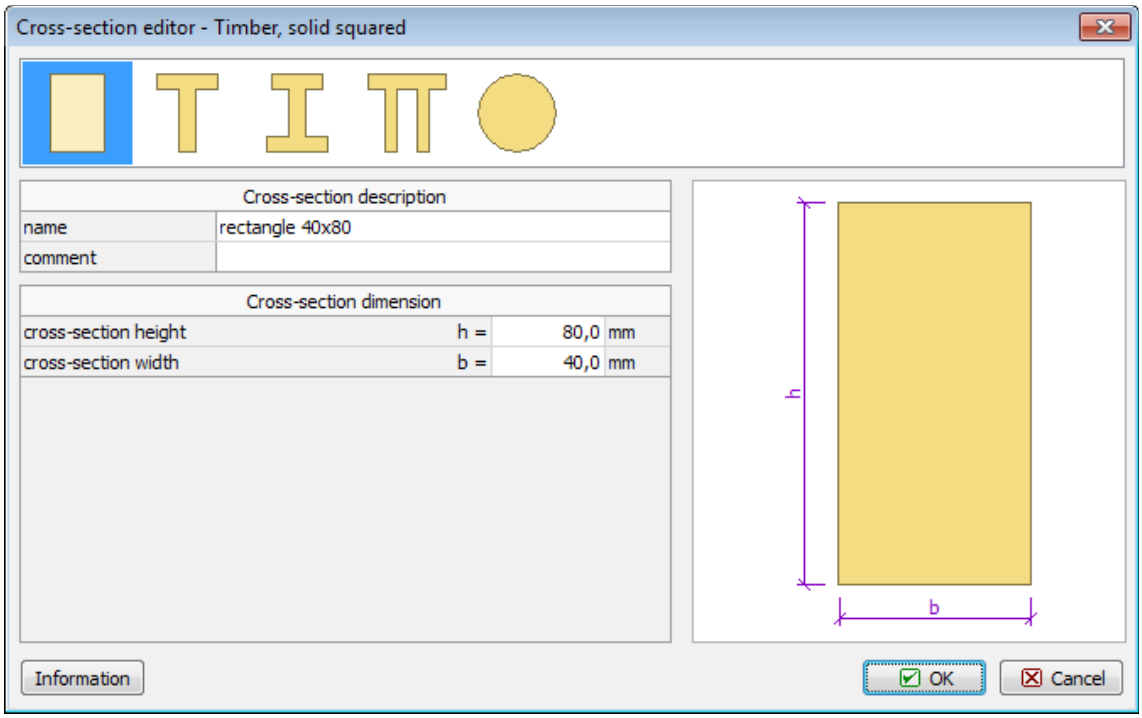
After definition of the truss dimensions, we need to specify cross-sections and materials of individual truss members. This will be done in the following dialog box "**Profiles in groups**" in which we can specify profiles and materials separately for the upper and the bottom chords and the filling studs by clicking the "**Profile**" button in the relevant tabs. However, we will use a quicker approach specifying first the material and profile for all members by clicking the "**Global profile**" and then only changing the profiles of the upper and the bottom chords.



"Profiles in groups" dialog box

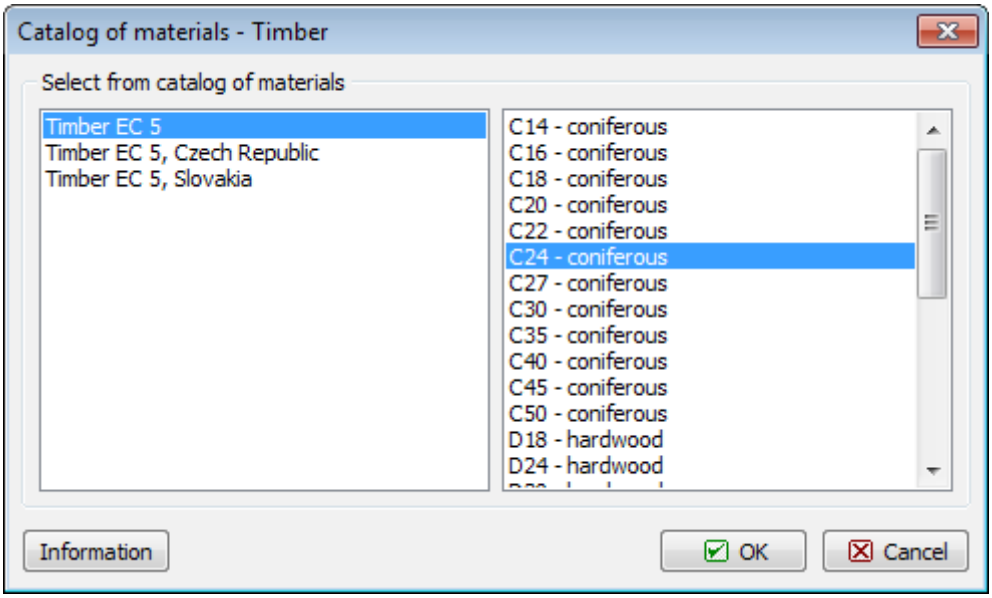
After clicking the "**Profile**" button, the "**Cross-section editor**" dialog box appears on screen. In this dialog box we select "**Timber**" and "**Solid squared**" type and define the dimensions of

the rectangular profile as *h = 80 mm* and *b = 40 mm*.



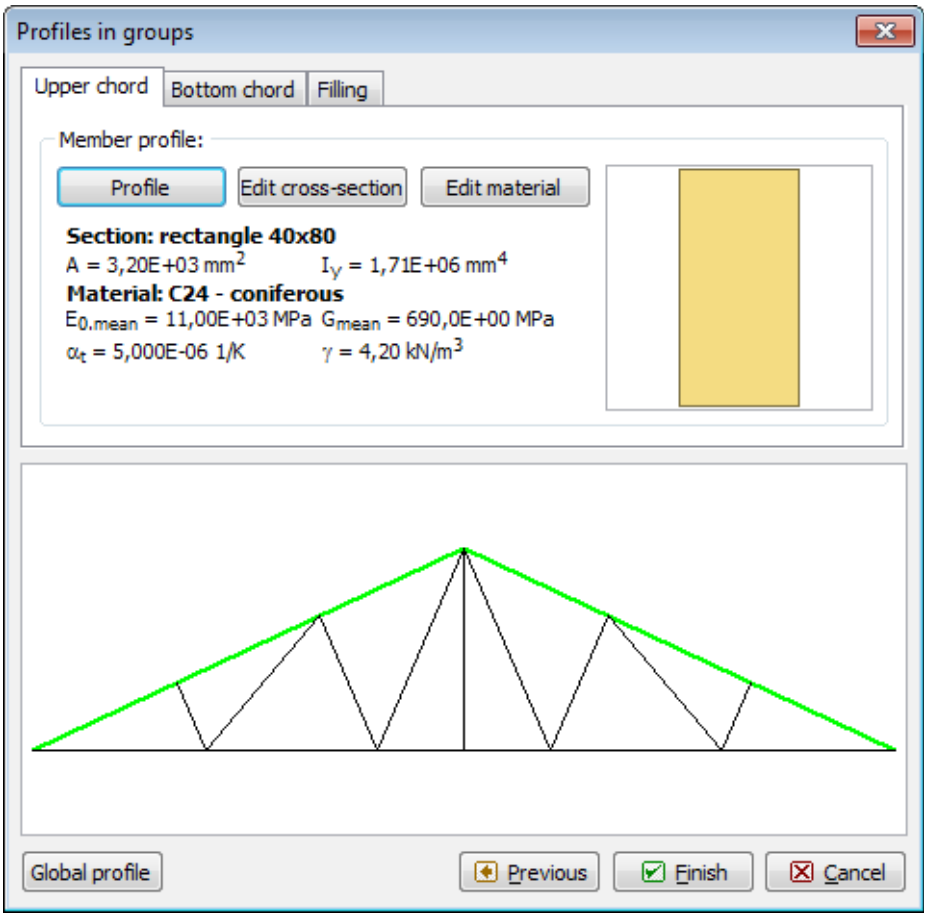
Defining cross-section dimensions

After confirming the dimensions by clicking "**OK**", the "**Catalogue of materials – timber**" dialog box appears, offering a selection of the standard timber strength classes. We select **C24** and confirm by clicking the "**OK**" button.



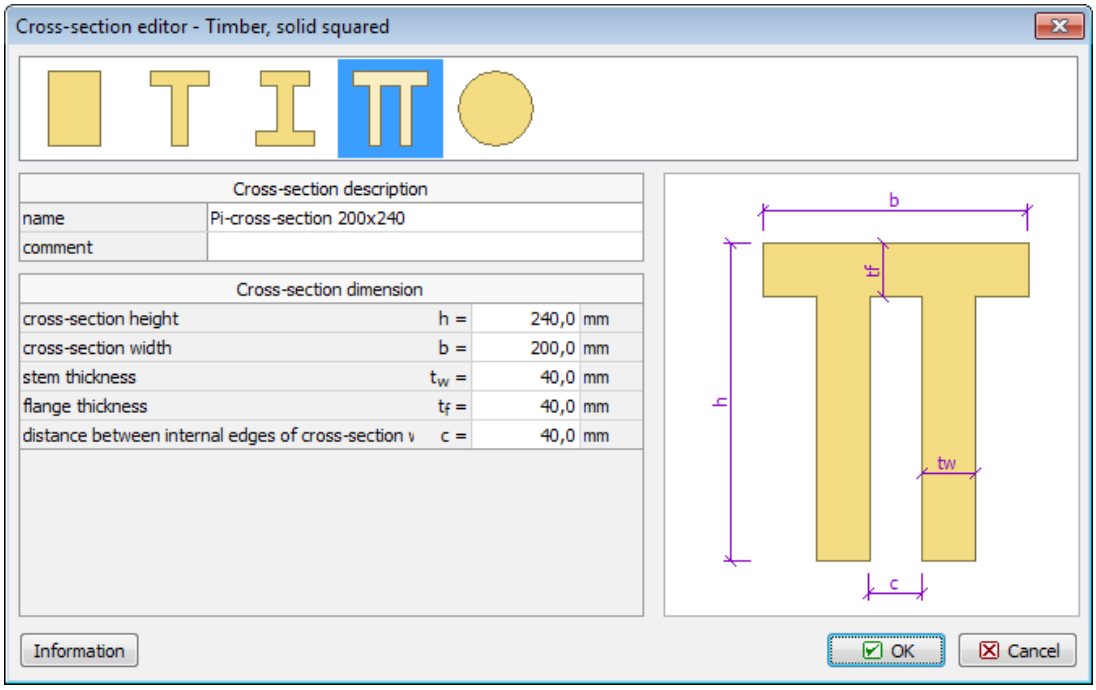
Selection of strength class

After confirming, the selected material and cross-section data appear in all three tabs of the "**Profiles in groups**" dialog box.



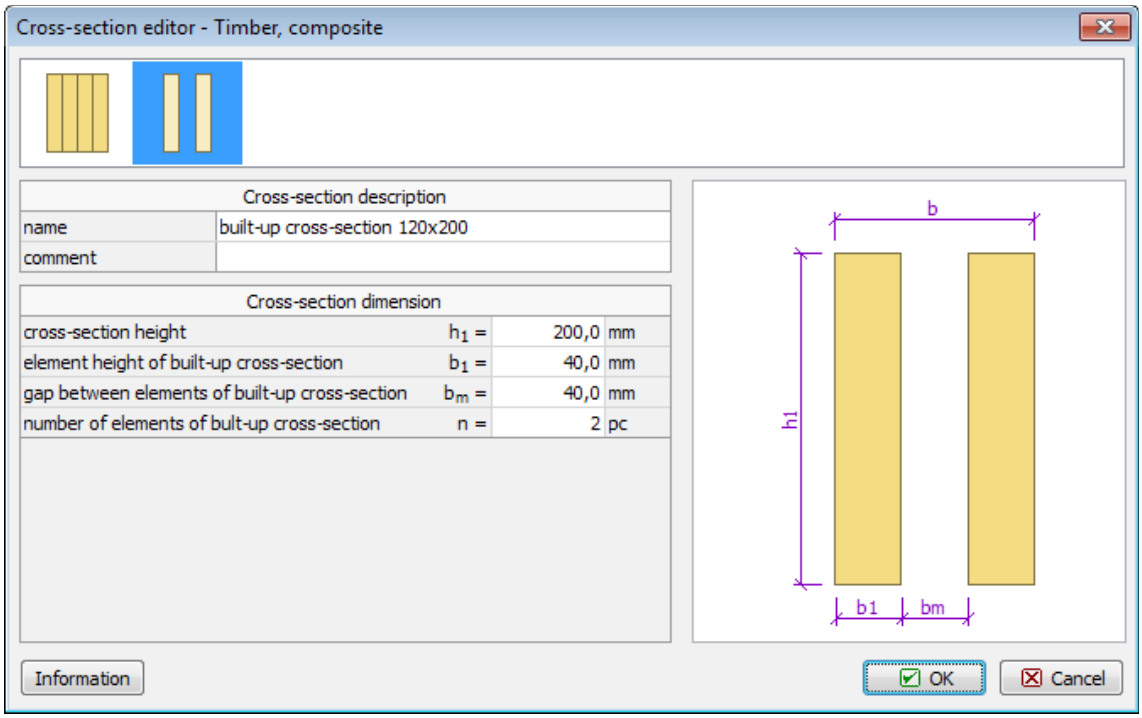
Editing profiles of upper and bottom chords

Now we change the profiles of the upper and the bottom chords. First, in the "**Upper chord**" tab using the "**Profile**" button, we set a Pi-shaped profile and enter the dimensions.



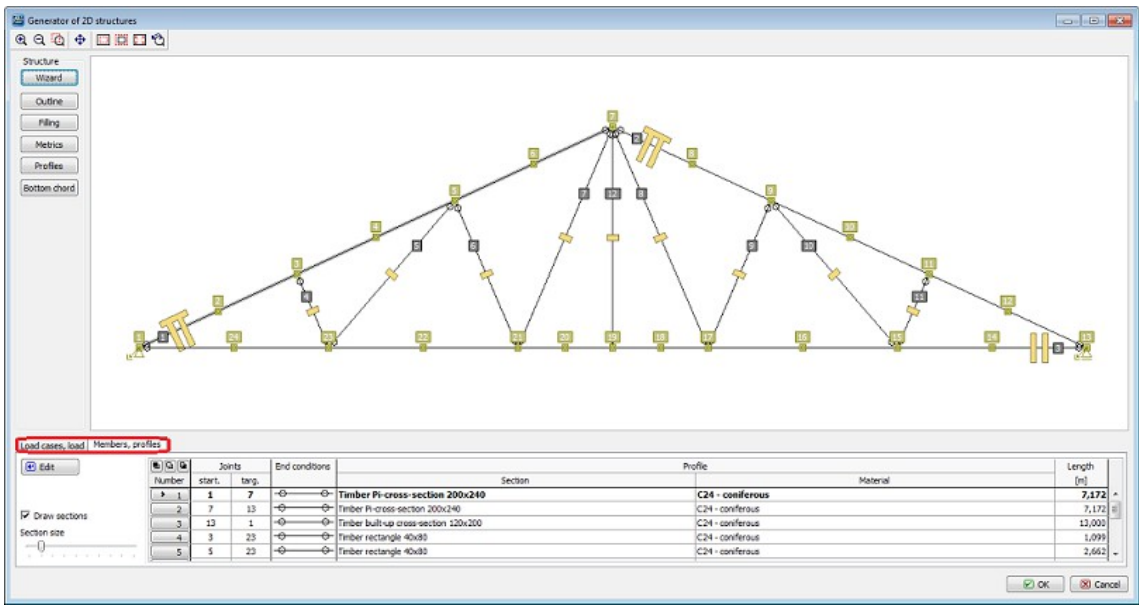
Upper chord's cross-section dimensions

Analogically, we define a compound cross-section of the bottom chord.



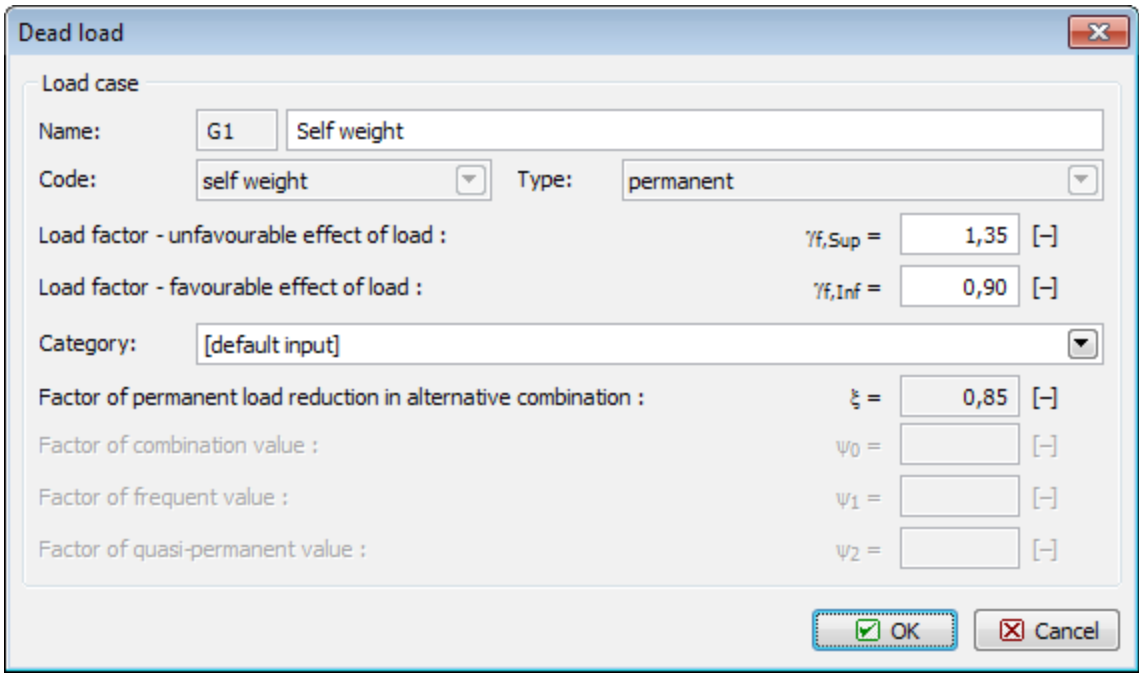
Dimensions of bottom chord's compound cross-section

After changing the chords' profiles we exit the "**Cross-section editor**" by clicking "**OK**". The generated truss is now displayed on screen. We can go back to any of the previous steps using relevant buttons in the "**Structure**" frame to the left from the model space. In the frame in the bottom of the screen, the tables for managing load cases and members are organized into tabs.



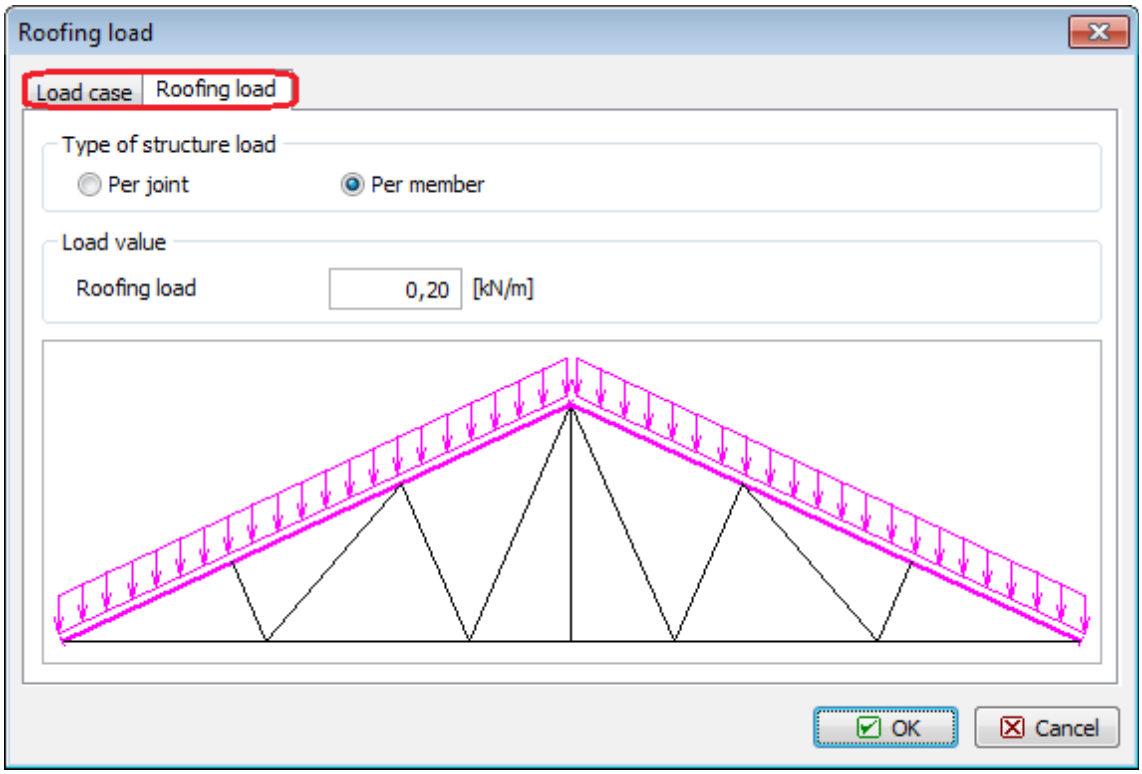
Tabs for editing load cases

We select the "**Load cases, load**' tab and begin to define the load cases. Firstly, selecting "**Self weight**" we define a load case, which will contain automatically generated loads from the truss's self weight. In the dialog box we can edit the name of the load case or load factors.



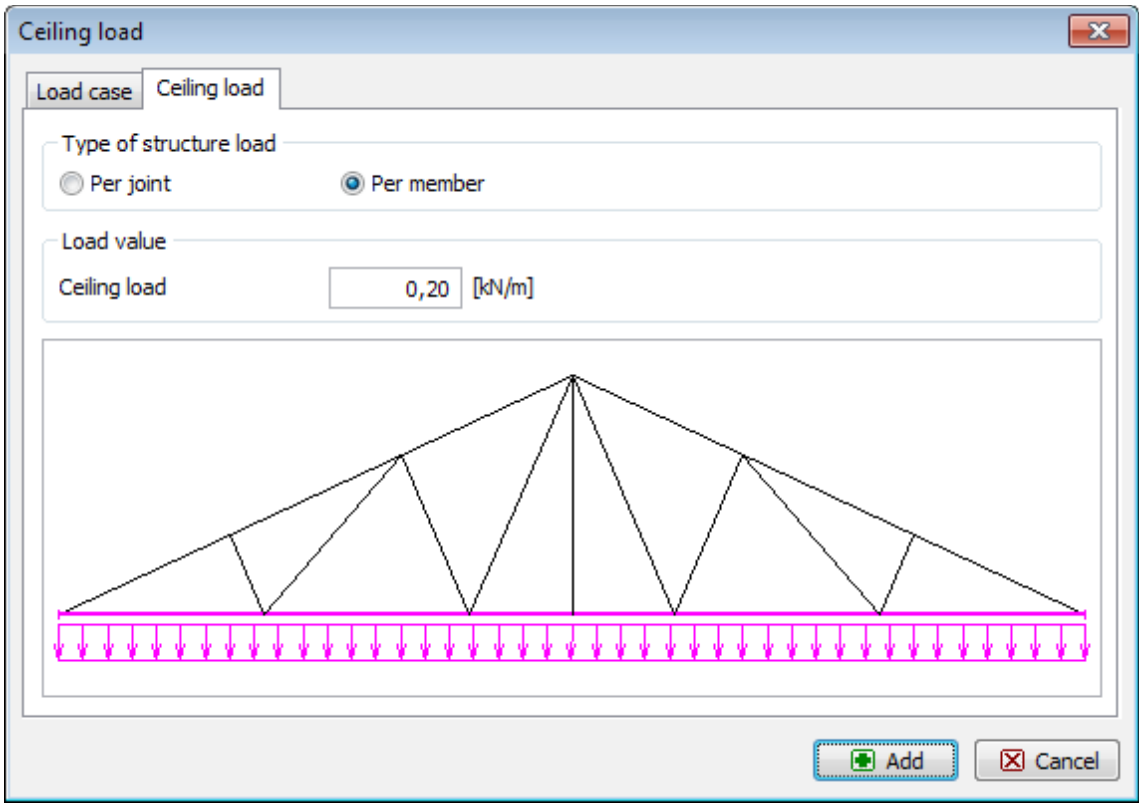
Parameters of "self weight" load case

Adding a new load case is confirmed by clicking "**OK**" and the loads generated from the self weight into this load case are instantly displayed in the model space. We continue with adding loads from the roofing, using the "**Roofing**" button. The "**Roofing load**" dialog box contains two tabs; the first is for specifying the load case parameters (similarly to self weight), in the second the load magnitude is defined. We switch to the second tab to enter the value *0.2 kN/m* and add the load case by clicking "**OK**". Then we exit the dialog box by clicking the "**Cancel**" button.



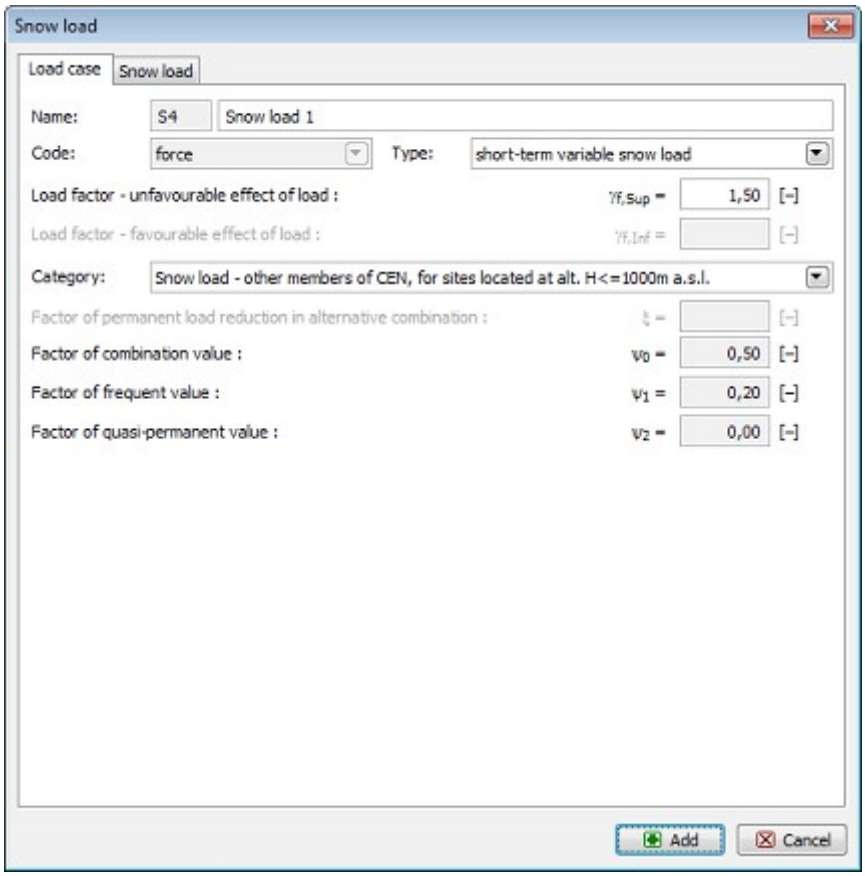
Tabs in the 'Roofing load' dialog box

We repeat the same procedure to define the ceiling loads.



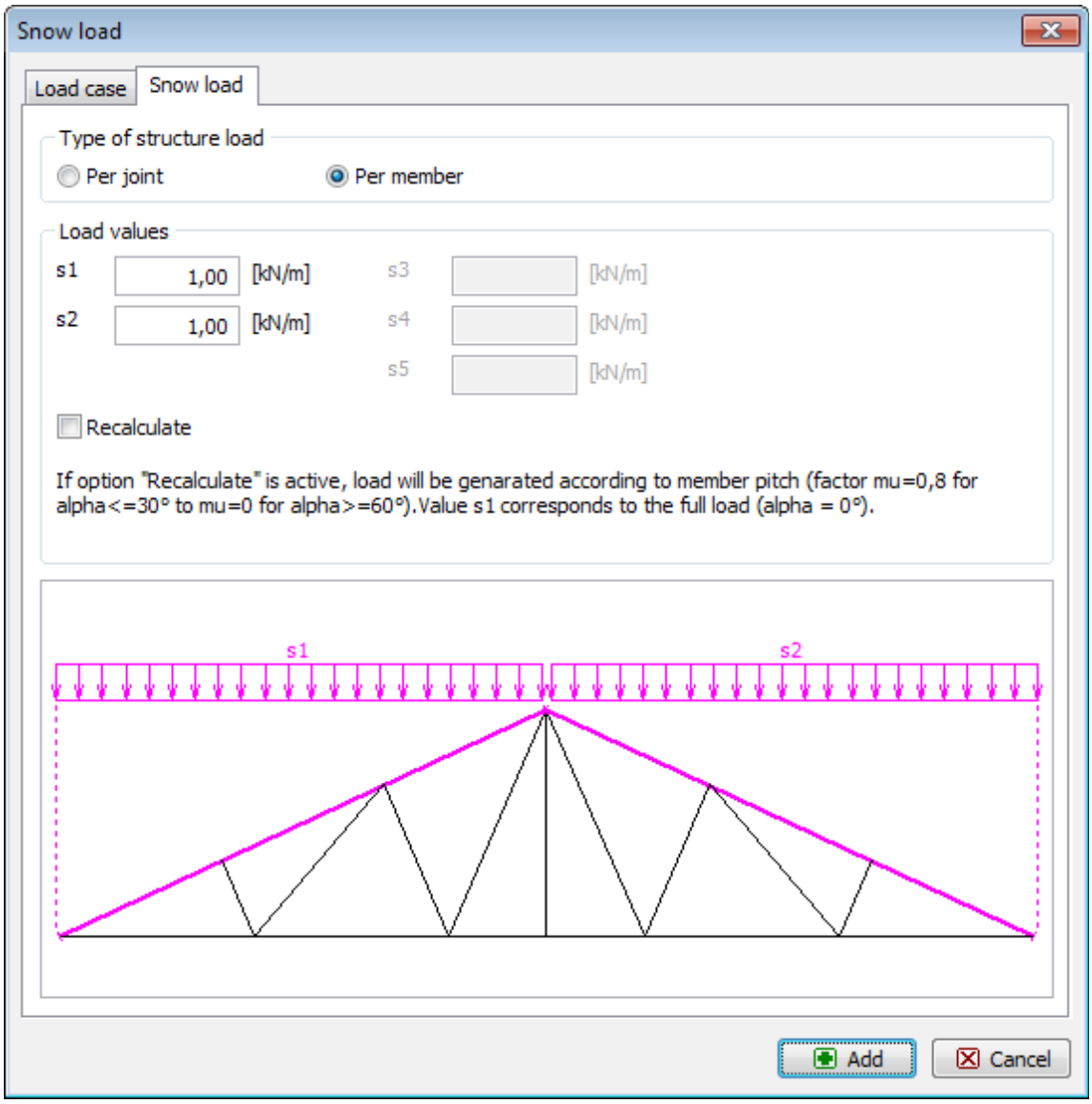
Defining ceiling loads.

For snow loads, due to the variable nature of the loading, the dialog box for the load case properties contains different data than that for the permanent loads. Short or medium term loading type can be selected, as well as the "**Category**" which sets the combination factors in accordance with EN 1990.



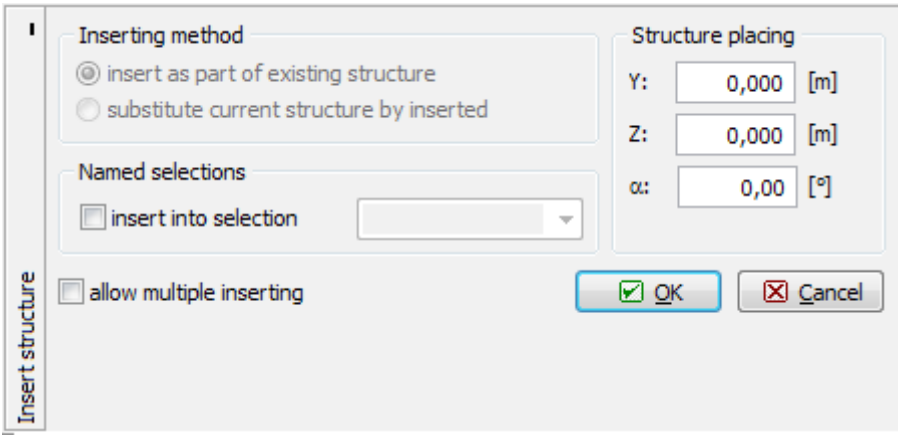
Snow load properties

In the second tab, we can define the loads separately for the left and the right side of the truss; it is possible to define non-uniformly distributed load caused by snowdrifts. The magnitude of the loading can be entered as the basic value obtained from the snow map; automatic redistribution on the inclined plane is run by ticking the "**Recalculate**" box. First we define the load case with uniformly distributed loads *1.0 kN/m* applied to both halves of the truss; the load is applied to the structure by clicking the "**Add**" button. Then we can change the value *s1* to *0.5 kN/m*; thus we obtain a non-uniformly distributed load case which we again apply to the truss by clicking the "**Add**" button. Finally, we switch the values *s1* and *s2* to obtain a load case symmetric to the previous. We apply it by clicking the "**Add**" button and the "**Cancel**" button to exit the dialog box.



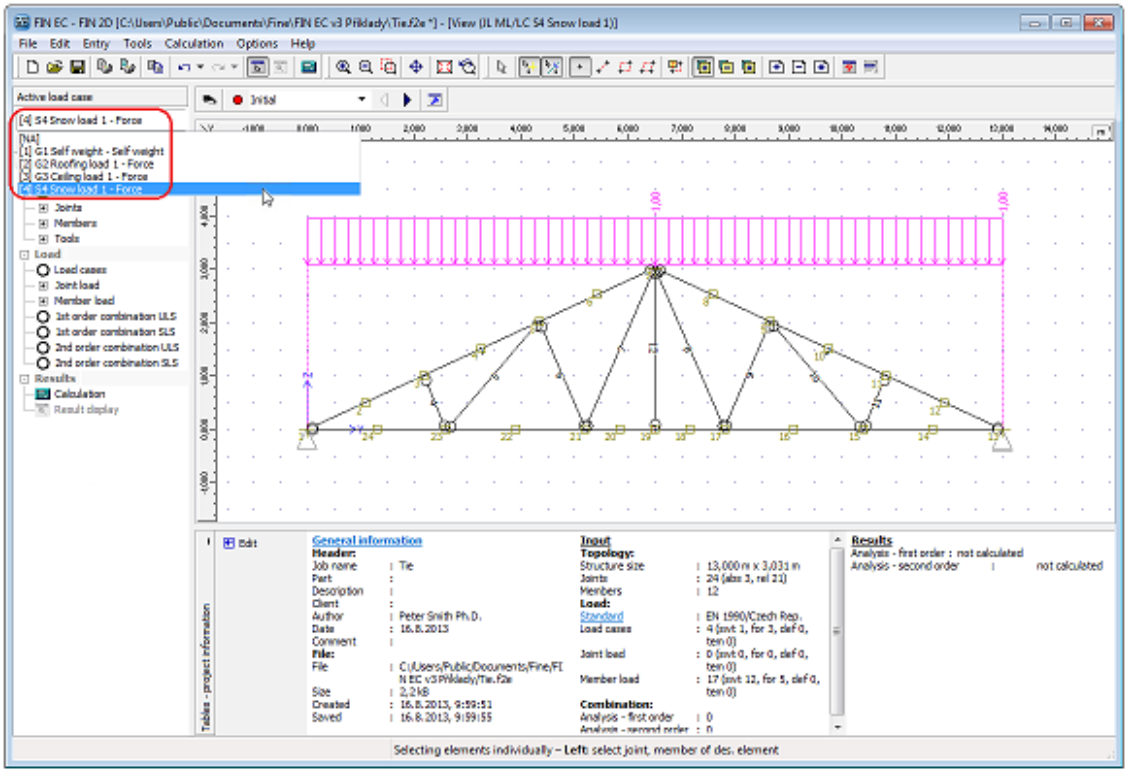
Defining snow loads

We can check and amend the defined load cases using the table in the bottom part of the Generator of 2D structures. If the load cases are correctly defined, we can insert the generated structure into the 2D program by clicking "**OK**". We can define structure placing and rotation in the table located in the bottom part of the main screen.



"Insert structure" table.

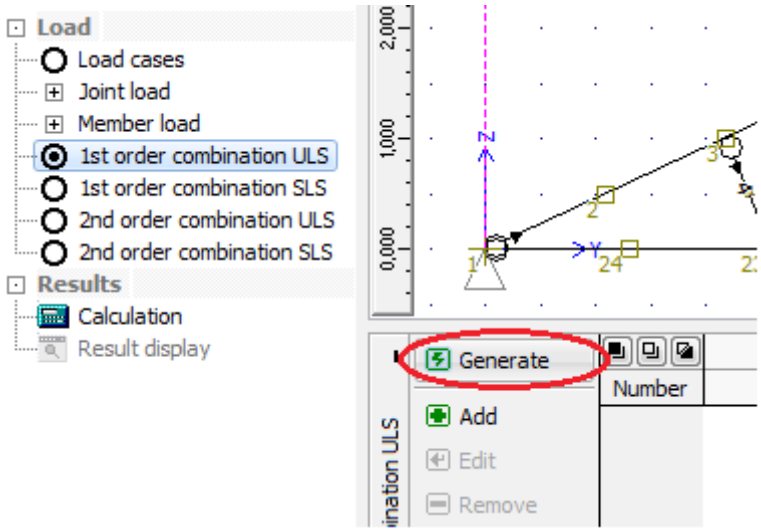
After inserting, the truss is displayed in the program's model space. Geometry can be further edited in the "**Topology**" part of the control tree, load cases and loads can be edited in the "**Loads**" part. Only the active load case selected from the drop down list in the upper part of the control tree is displayed in the model space.



Displaying particular load cases

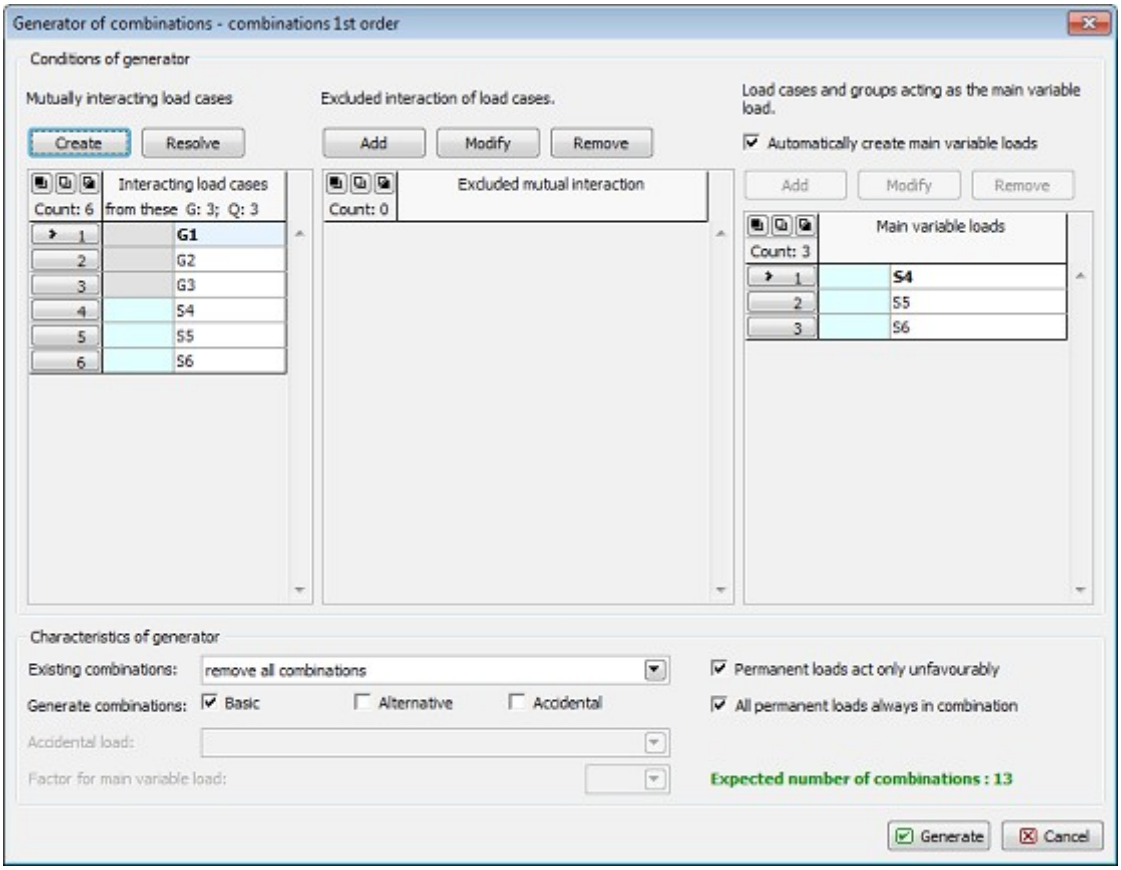
Definition of combinations

We can proceed to defining the load combinations which are defined separately for the ultimate and the serviceability limit states. First we define the combination for the ultimate limit state. We switch to "**1st order combination ULS**" in the control tree and run automatic definition of combination by clicking the "**Generate**" button in the table of combinations.



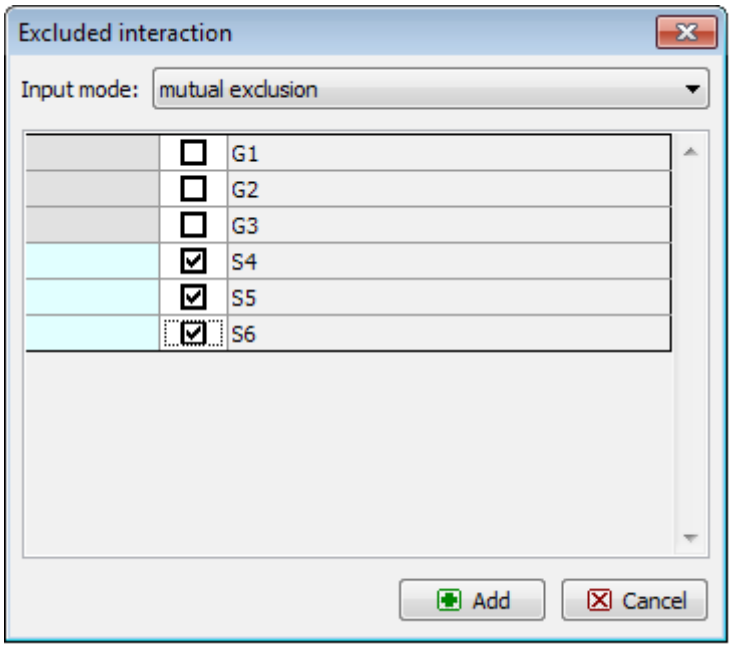
Button for automatic definition of combinations.

The automatic generation of combination is run in the "**Generator of combinations**" dialog box. The dialog box contains three tables. In the first, the load cases which act simultaneously are combined. In the second, we can set mutual exclusion of some load cases in one combination. The last table contains a list of variable loads which shall be considered main. In our example, we need to exclude simultaneous action of the defined snow load cases; hence we create a new exclusion group by clicking the "**Add**" button in the table "**Excluded interaction of load cases**".



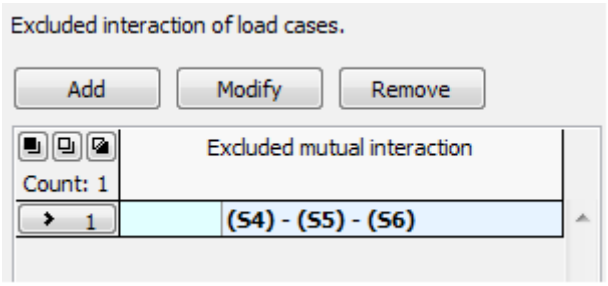
"Generator of combinations" dialog box.

In the "**Excluded interaction**" dialog box we select the load cases *S4*, *S5* and *S6* and confirm selection by clicking the "**Add**" button.



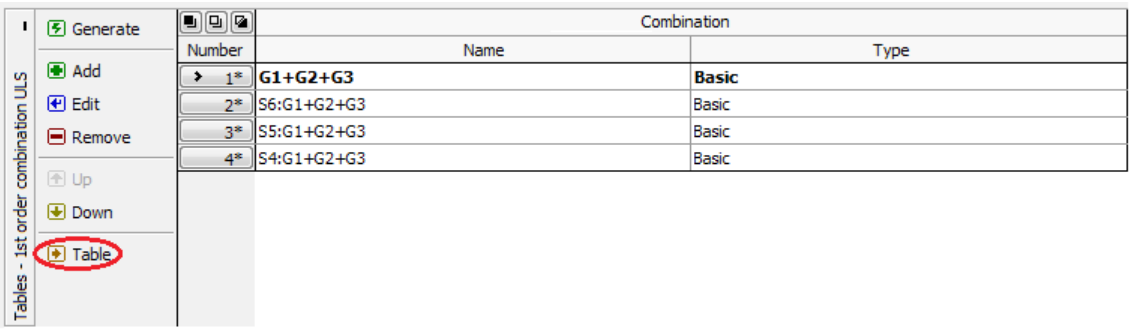
Defining mutual exclusion of load cases

After closing the dialog box a new group of mutually exclusive load cases appears in the relevant table. Thus it is guaranteed that only one of these load cases can appear in one combination.



Added group of mutually exclusive load cases

Once we have finished entering data, we can create the combinations by clicking the "**Generate**" button. A list of the generated combinations appears in the table in the bottom part of the screen; we can add, edit or erase the combinations as necessary. We can also display the list in a comprehensive table by clicking the "**Table**" button.



Button for running "Table of combinations"

In the "**Table of combinations**" we can check the generated combinations; for the active combination, a detail description including the used load factors is displayed in the bottom part of the table.

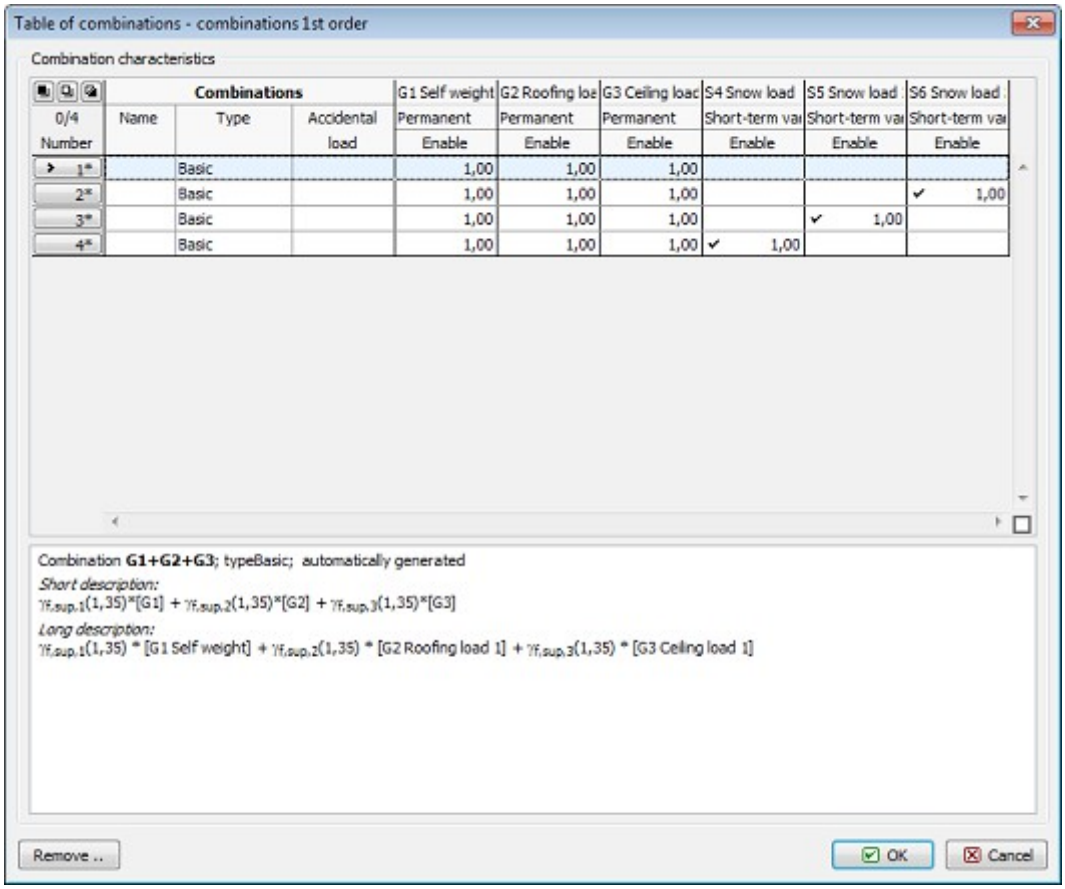
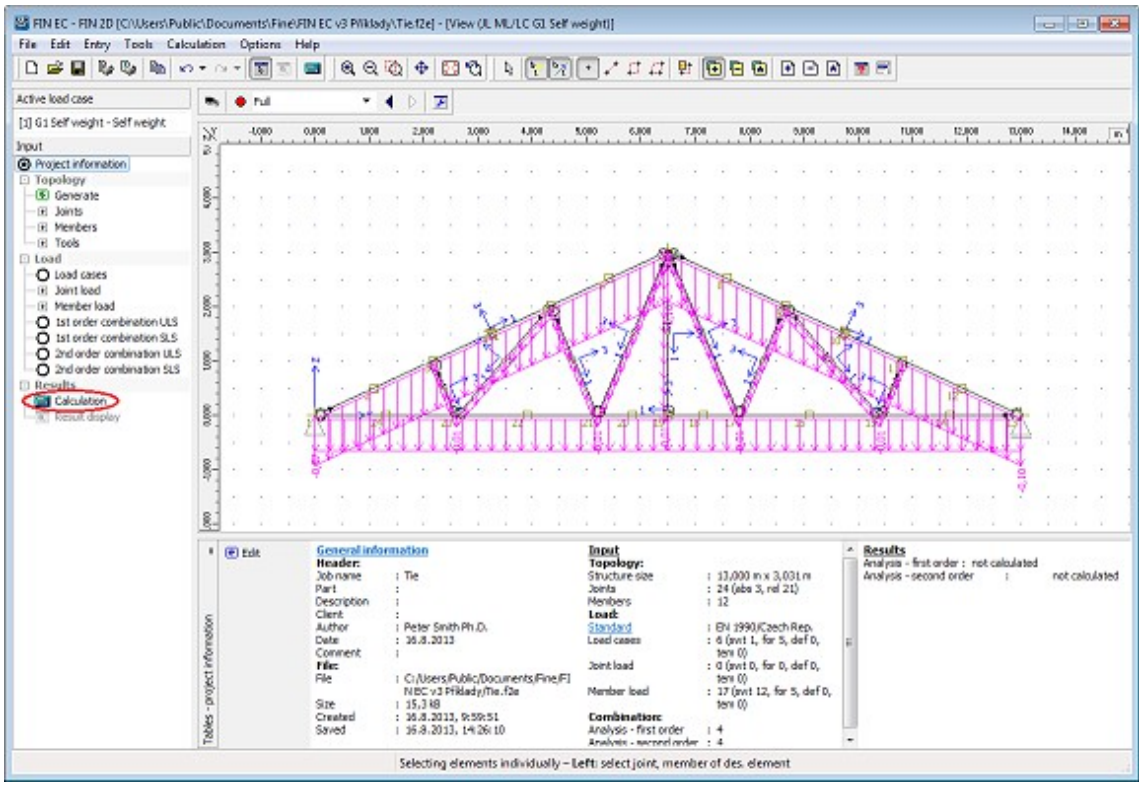


Table of combinations

Analogically we will generate the characteristic combinations switching to "**1st order combinations SLS**" in the control tree.

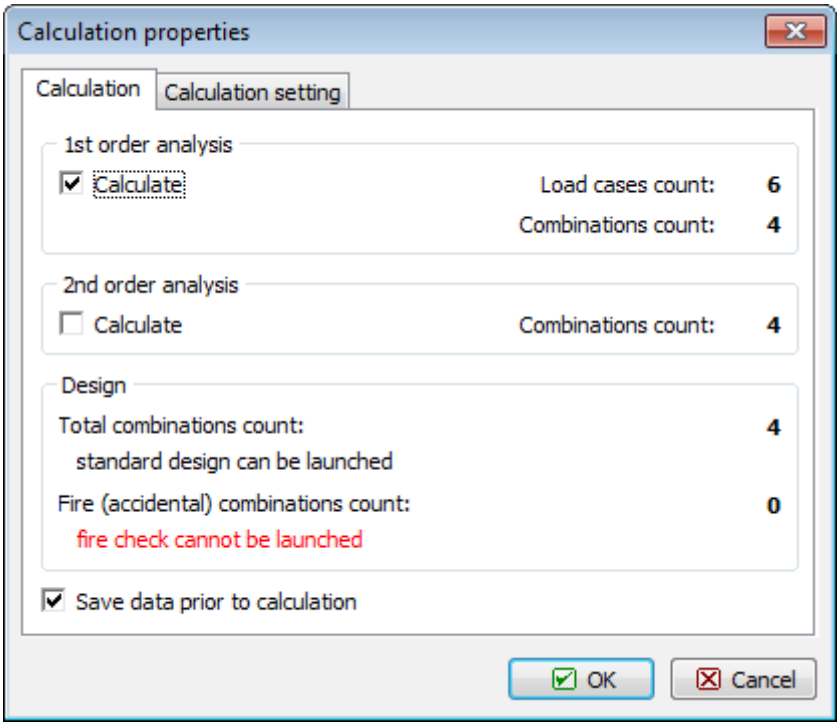
Calculation and results display

Now we can finally proceed to running the "**Calculation**" of internal forces by clicking the eponymous button in the control tree.



Running calculations

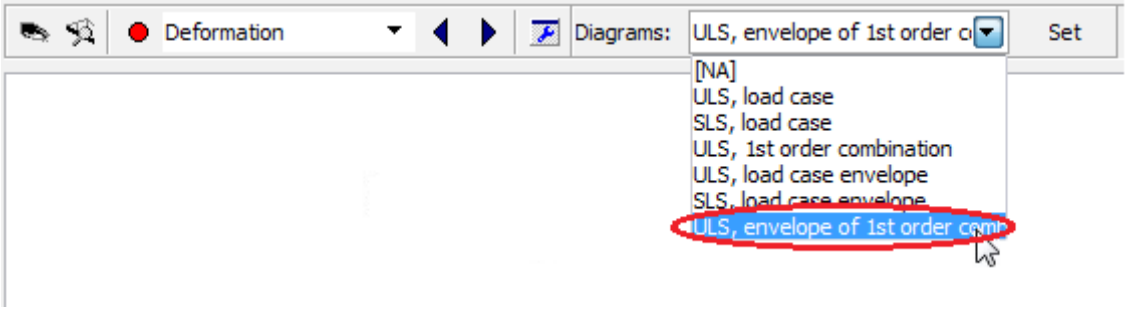
The "**Calculations properties**" dialog box appears; we can confirm the settings by clicking "**OK**" after which the calculation is executed and a window with information about the calculation process displayed. After clicking the "**Cancel**" button, program switches automatically to post-processor.



"Calculation properties" dialog box.

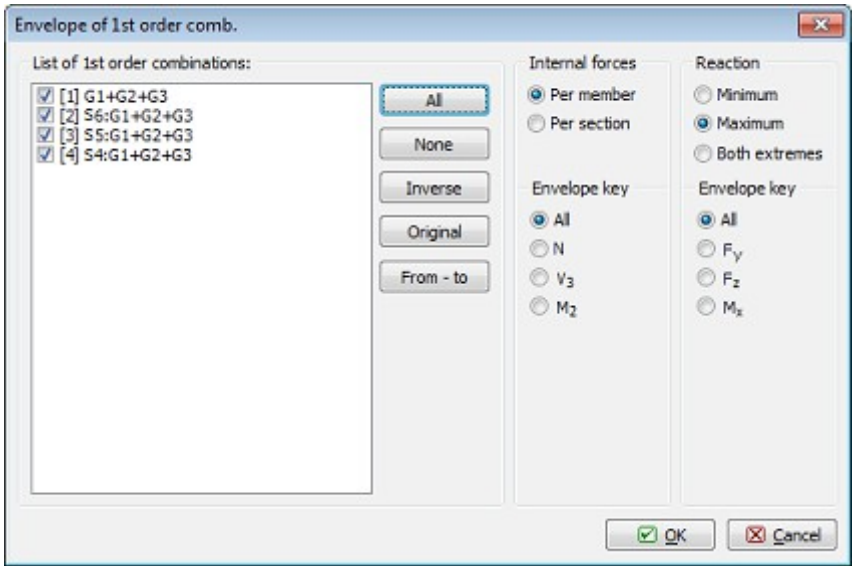
After finishing calculation, deformation resulting from the combination *No. 1* is displayed in the

model space. The program offers, apart from many other functions, a variety of settings for results displaying, e.g. enables saving views into the "**Named selections**" and printing all views subsequently. In our example we will show how to display the envelope of the bending moments. First we select "**ULS, envelope of 1st order combinations**" in the drop down list.



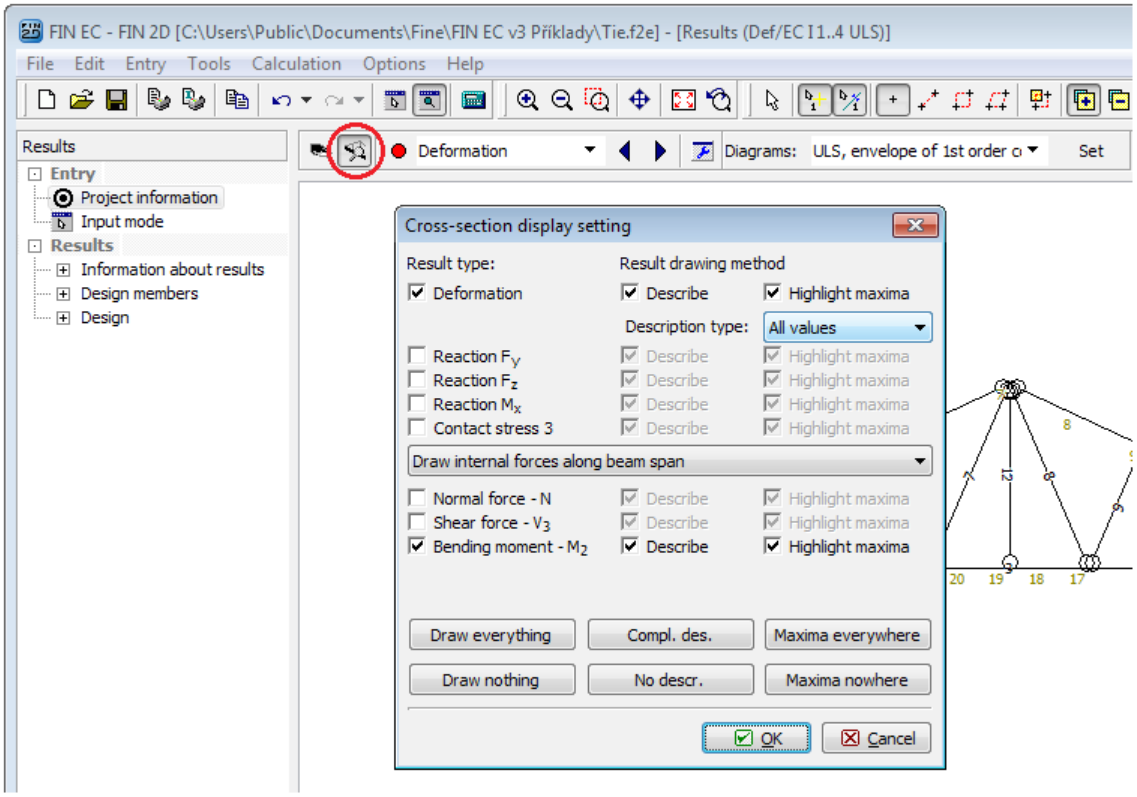
Selecting envelope display

To define an envelope of all combinations we click the "**All**" button in the dialog box; thus all combinations in the list on the left-hand side are automatically selected.



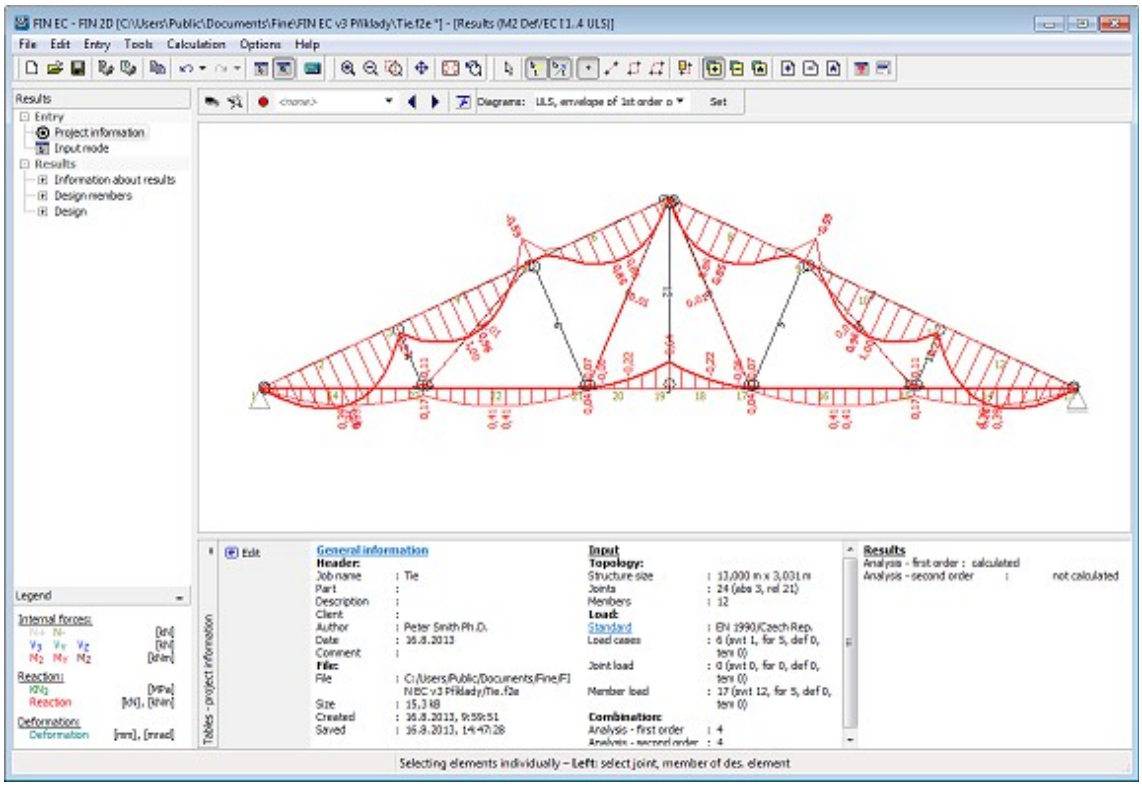
Selecting combinations for envelope

Then we run the "**Cross-section display settings**" dialog box and select "**Bending moment**".



"Cross-section display setting" dialog box.

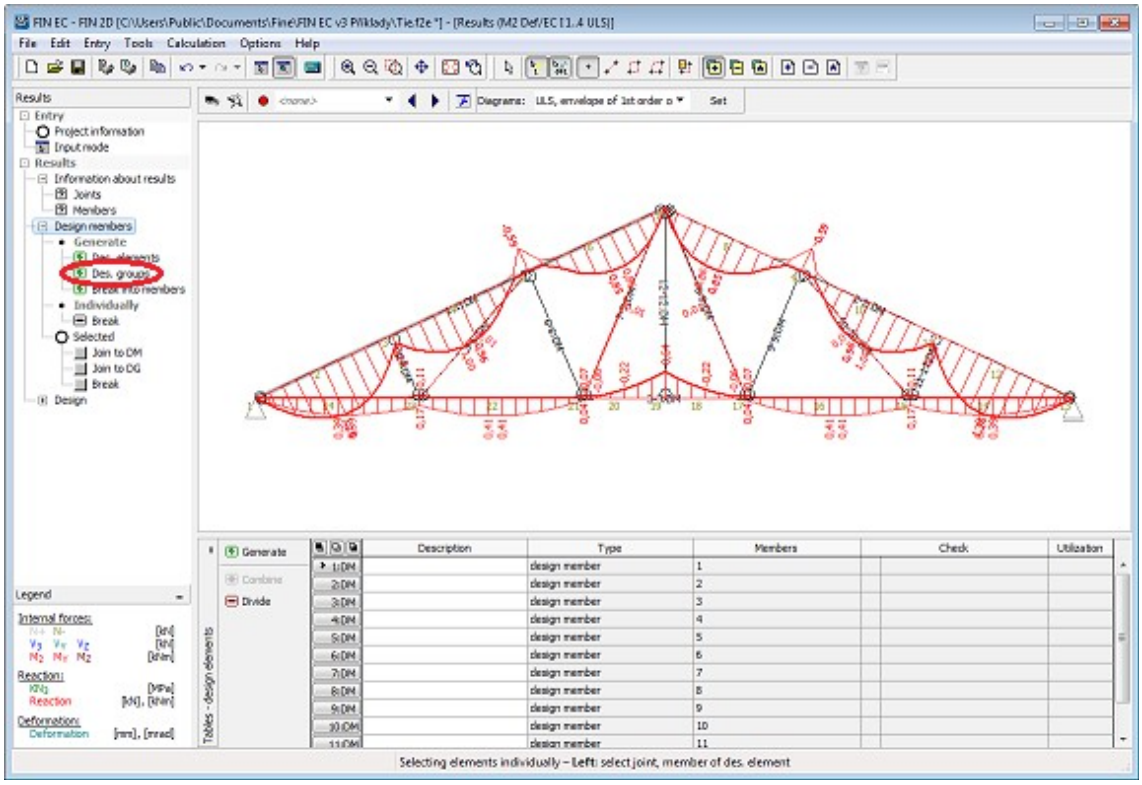
After confirming, the envelope of bending moments is displayed on the structure.



Envelope of bending moments on structure.

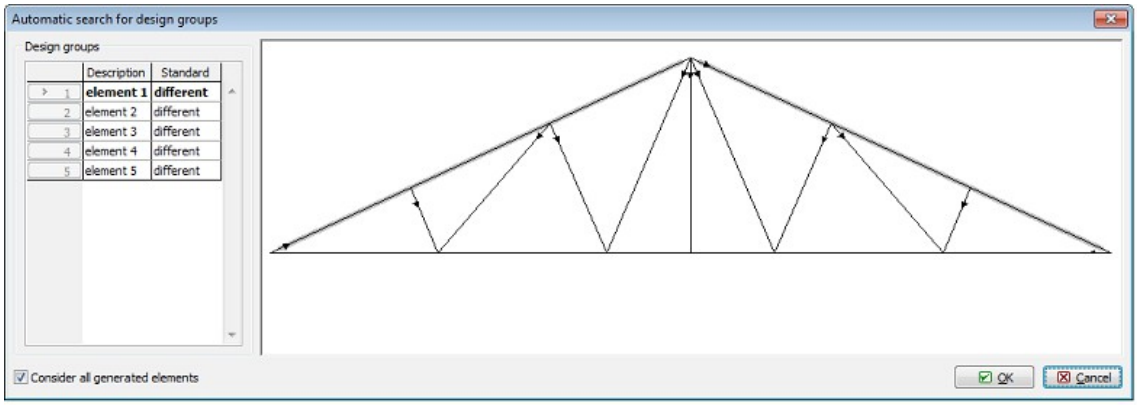
Now we proceed to design checks of the structure's cross-sections. First we switch to the "**Des. groups**" in the control tree. The structure consists of total of 11 elements, representing 11

design members. The program enables merging the members into design groups so that the assessment is as quick and straightforward as possible. Members merged into a design group are checked as one member; the loading is however considered on all members separately. This approach is beneficial for instance in case when we need to check a number of concrete columns in which we want to have unified reinforcement – it is sufficient to merge all of them into one design group. To create design groups automatically, we select "**Generate – Design Groups**" in the control tree.



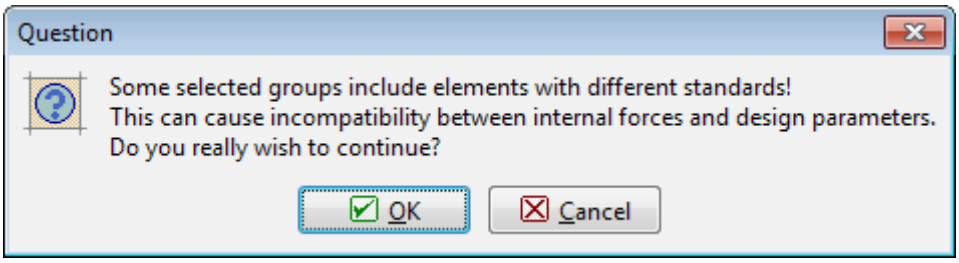
Generating design groups

We can check in the dialog box which design groups were found by the program. If we want to create only some of the suggested groups, we can untick the "**Consider all generated elements**" box and then proceed only with selected design groups. In our example we will use all suggested groups, therefore we close the dialog box by clicking "**OK**".



Suggested design groups

Because the orientation of particular elements vary we need to appreciate in case of which members this could cause difficulties – in our example it could be the upper chord. However; cross-sections and buckling parameters are constant along the length of the upper chords, therefore varying axes orientation should not influence the results of the assessment.


Question
Some selected groups include elements with different standards!
This can cause incompatibility between internal forces and design parameters.
Do you really wish to continue?
OK
Cancel

Notice on different element's orientation

Individual elements have been merged into 5 design groups and one design member. We can name the members and groups in the table in the bottom part of the screen.

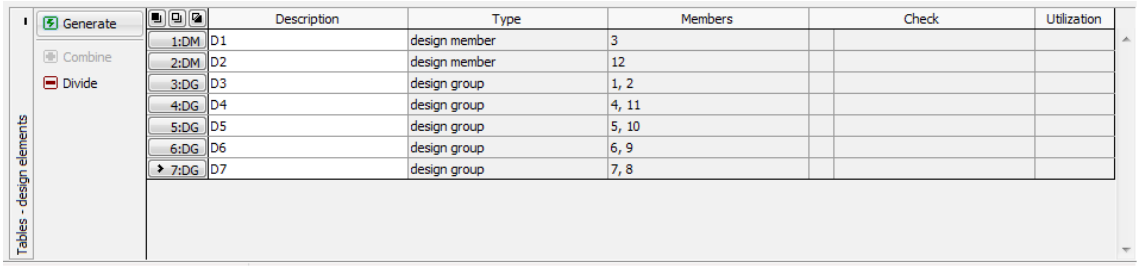
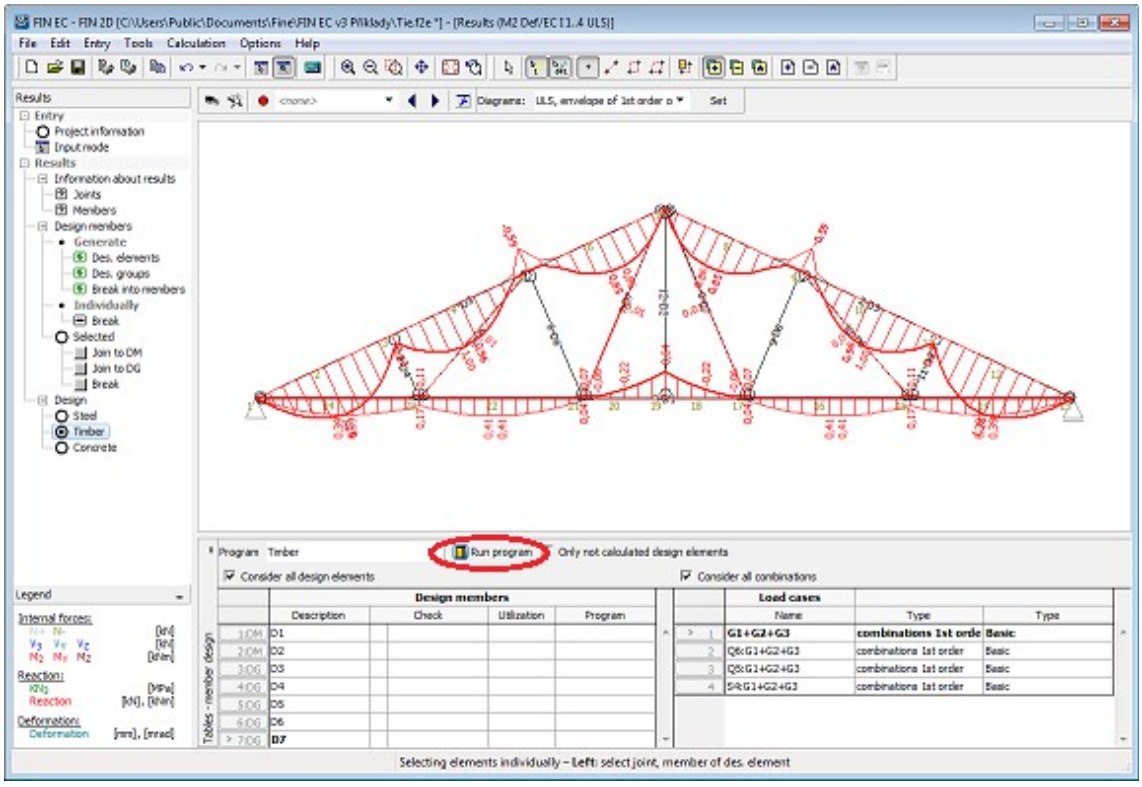


Table with entered names of design members and groups

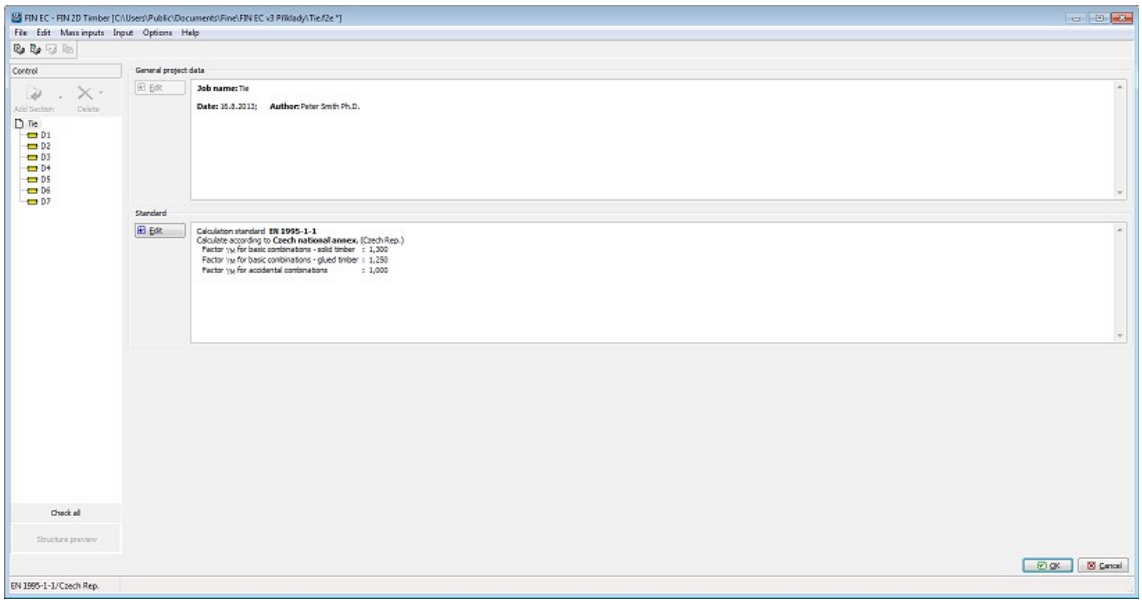
Now we can proceed to the design itself. We select "**Design**" and "**Timber**" in the control tree and run the timber structures design program by clicking the "**Run program**" button.



Running timber structures design program

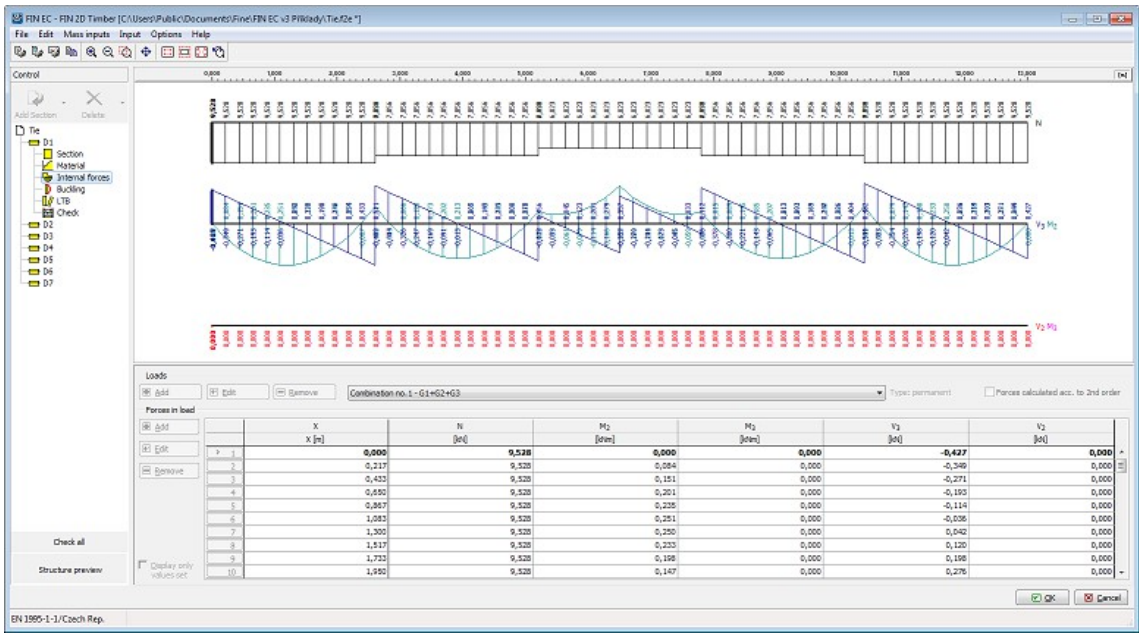
Members design

Program 2D Timber is run with all design members and groups automatically imported.



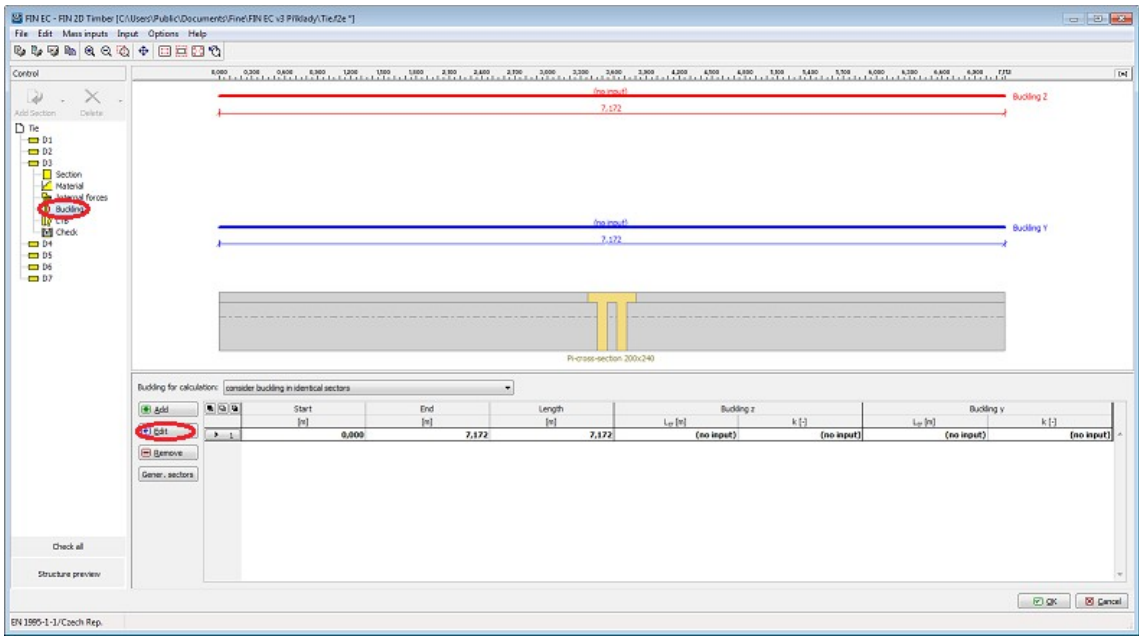
Design members in 2D Timber program

All data regarding geometry (members' lengths, cross-sections etc.) and loading (internal forces distribution for all combinations) have been imported into the program. The data can be checked in the relevant sections of the control tree. We can confirm position of a selected member in the structure by clicking the "**Structure preview**" button.



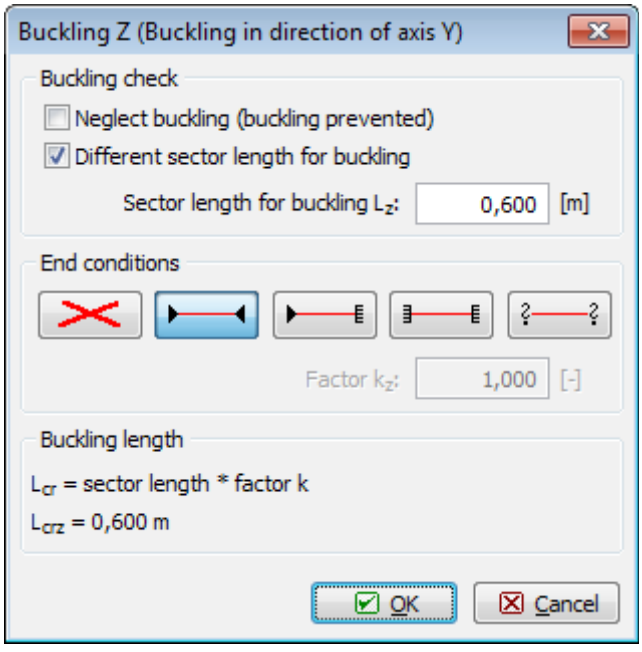
Distribution of internal forces in bottom chord

Proceeding to members design, we will demonstrate the procedure on the *upper chord* i.e. "**D1**" design group. The upper chord is subject to compression; therefore it is necessary to define the buckling parameters. In our example we assume that out-of-plane buckling is restrained by purlins at *0.6m* centres. We switch to "**Buckling**" in the upper chords section of the control tree and run the buckling parameters dialog box by clicking the "**Edit**" button.



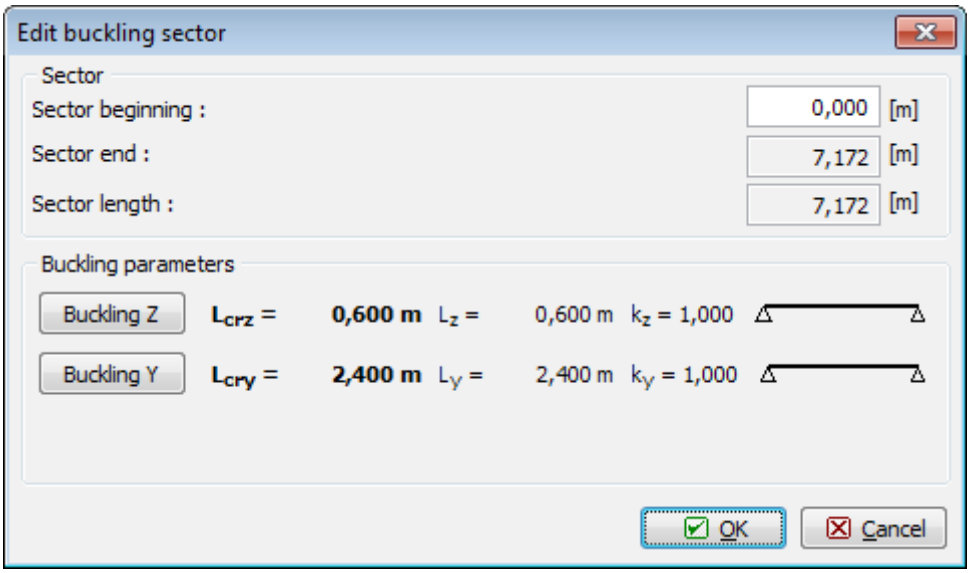
Editing buckling parameters

In the "**Edit buckling sector**" dialog box we can define parameters for out-of-plane buckling ("**Buckling Z**") and in-plane buckling ("**Buckling Y**"). For out-of-plane buckling we define simple end conditions and sector length for buckling $L_z = 0.6m$; for in-plane buckling we define simple end conditions as well with sector length 2.4 m. The parameters are defined in the "**Buckling Z**" dialog box.



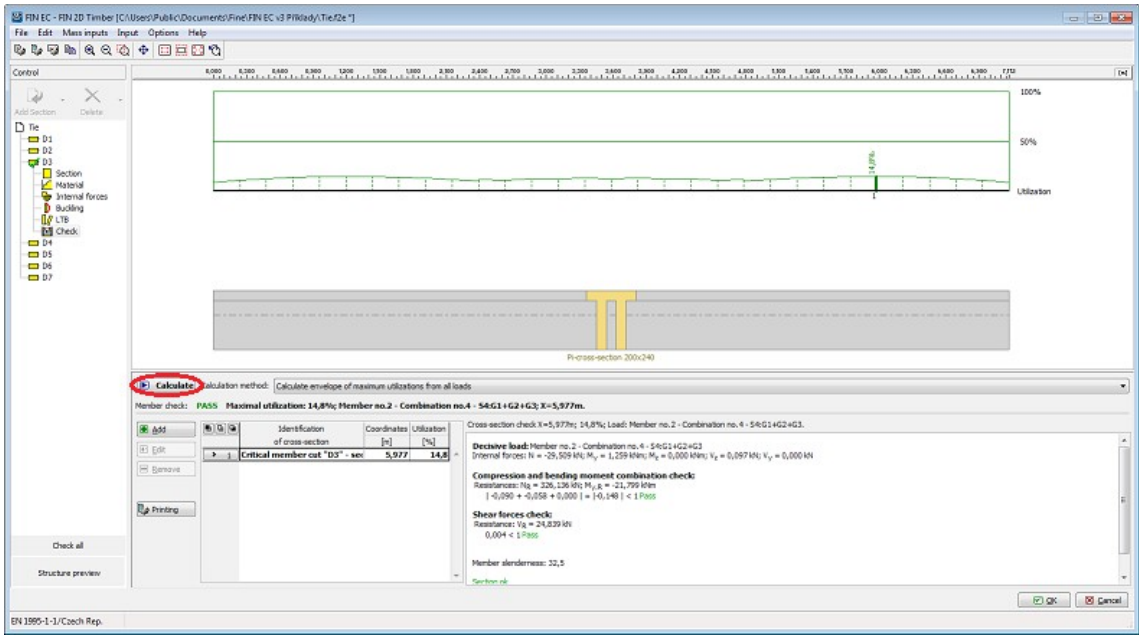
Defining in-plane buckling length

When the parameters are defined for both directions we can close the dialog box by clicking "**OK**".



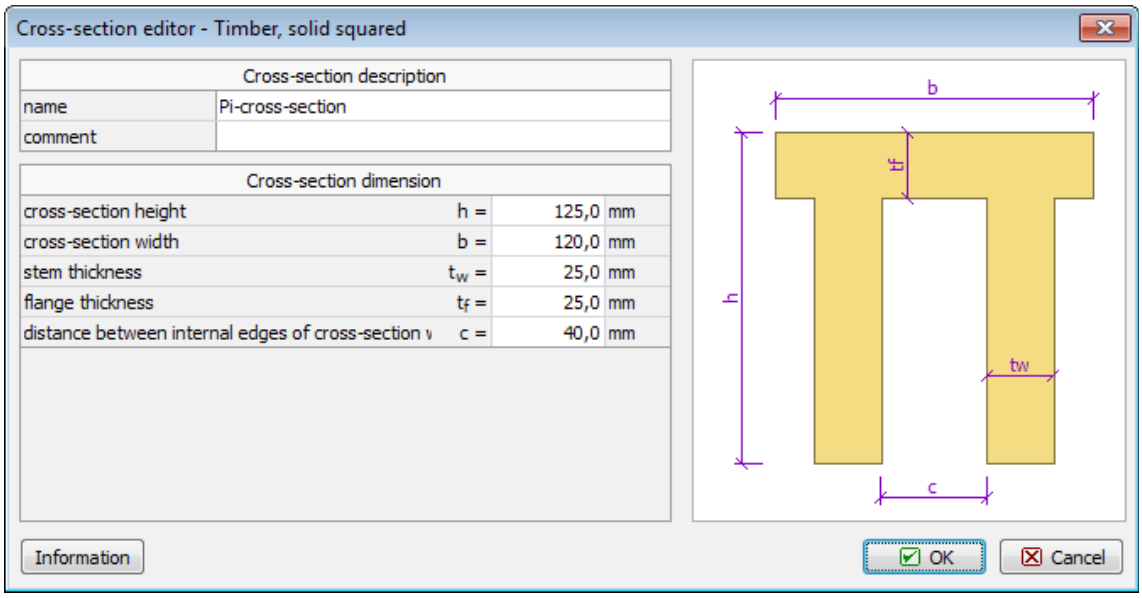
Defined buckling parameters

We switch to the "**Check**" section of the control tree and run the calculations by clicking the "**Calculate**" button. In the model space, the cross-section utilization curve is displayed along the length of the member; the critical section with the highest utilization is checked in the bottom right corner of the screen. If it is necessary to display detail checks in other sections, these can be added using the table in the bottom part of the screen or simply by double-clicking in the selected location of the utilization diagram in the model space.



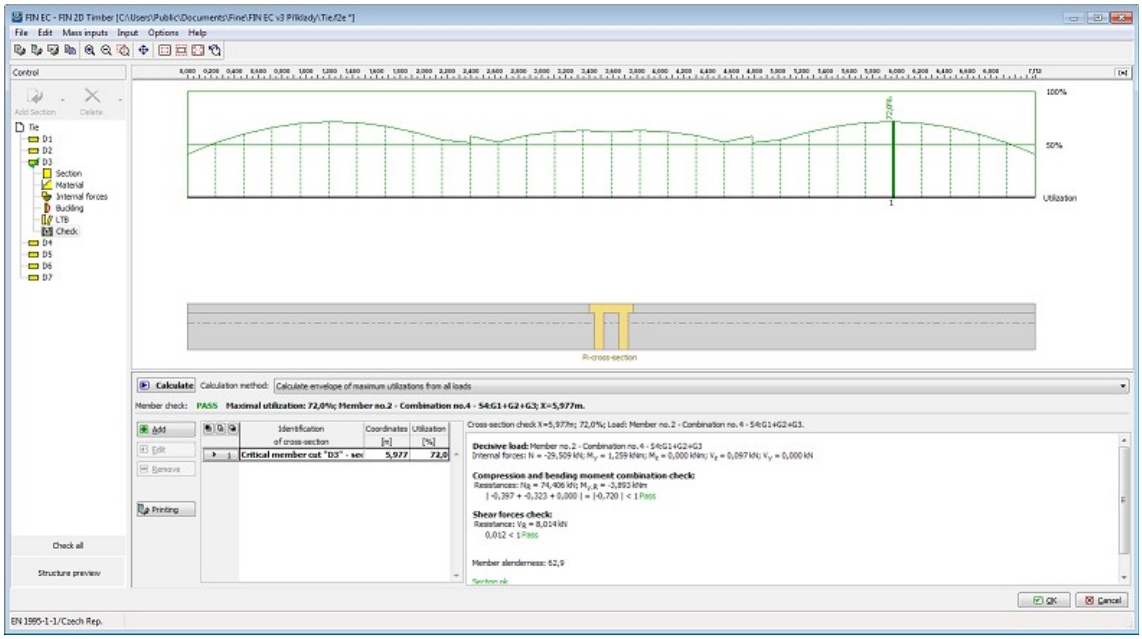
Members check

Because the maximum utilization of the member is very low, we can reduce the size of the cross-section. We switch to the "**Section**" part of the control tree and run the "**Cross-section editor**" dialog box where we can edit the cross-section geometry.



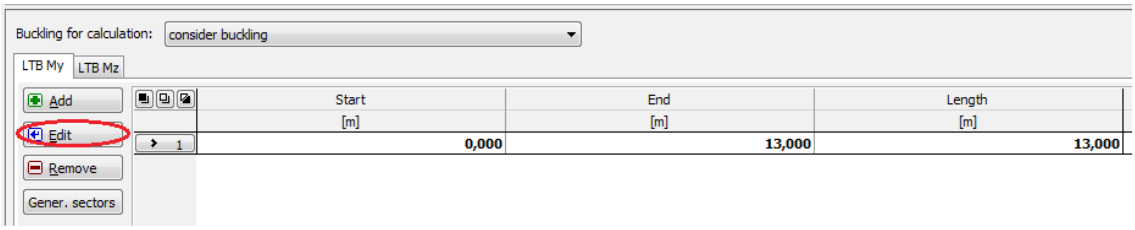
Edited dimensions of the upper chord

After editing the geometry as shown we return to the "**Check**" part of the control tree and re-calculate the structure, obtaining more acceptable check results.



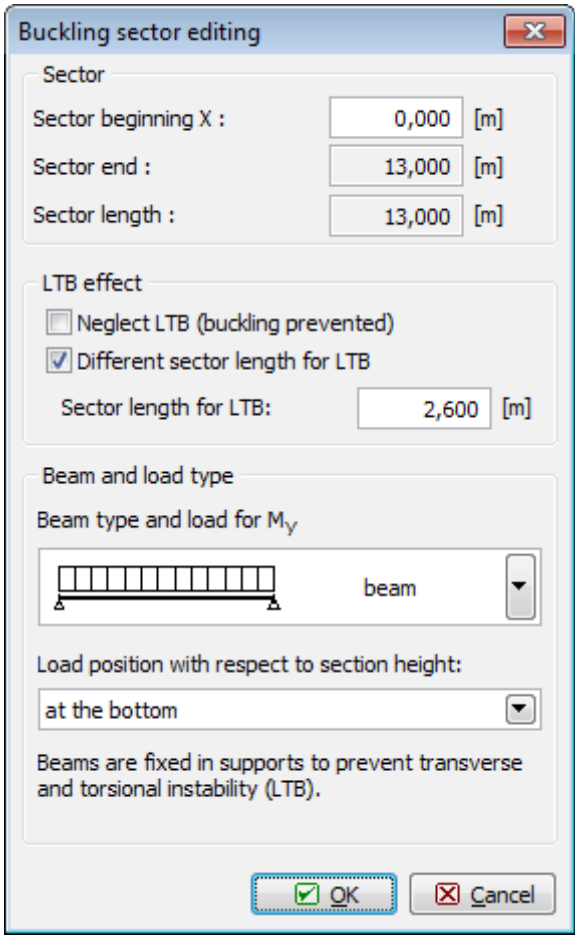
Optimized member check

Now we proceed with the "**bottom chord**". Because the bottom chord is in tension it is not necessary to define buckling parameters. However, we need to define lateral torsional buckling parameters as lateral and torsional stability should be checked in members subject to combination of tension and bending. We switch to the "**LTB**" section of the control tree and analogically to the buckling of the upper chord we define the buckling parameters for bending moment M_y by clicking the "**Edit**" button.



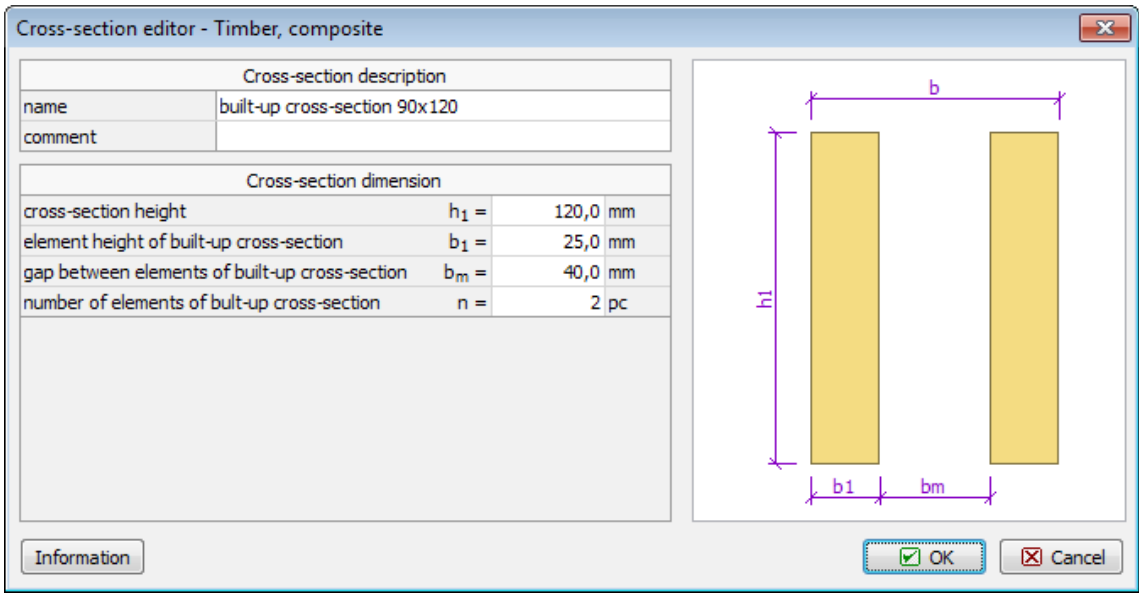
Editing buckling parameters

In the "**Buckling sector editing**" dialog box we define the LTB effective length and select appropriate beam and load type. We close the dialog box by clicking "**OK**".



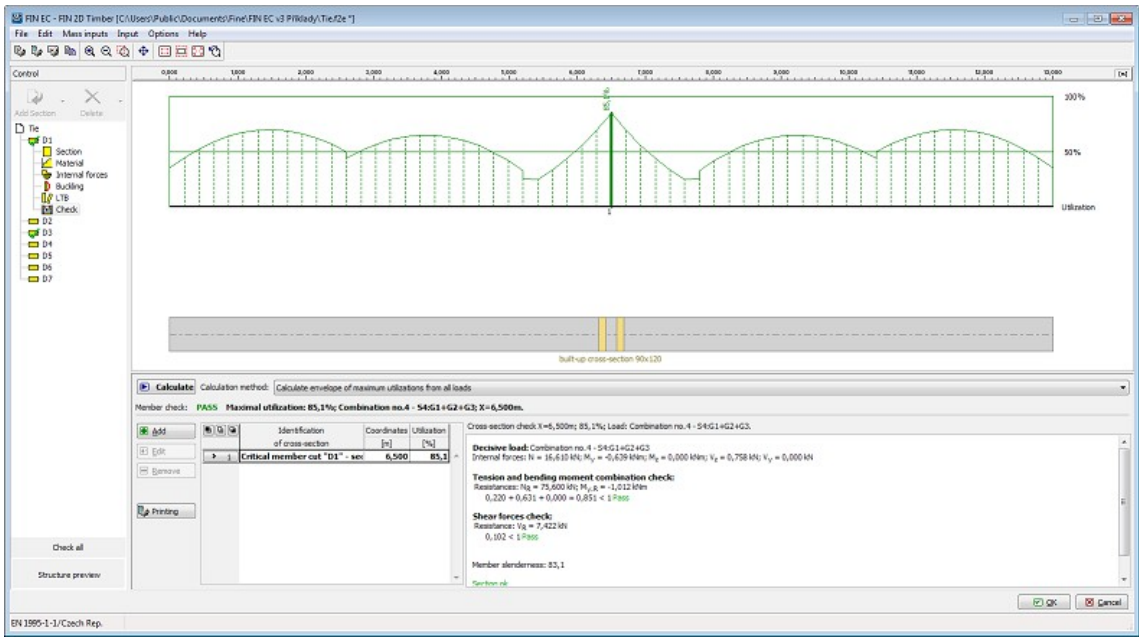
Buckling parameters

Then we switch to the "**Check**" section of the control tree again and carry out the members design check. Also for this member the utilization is too low therefore we edit the cross-section again.



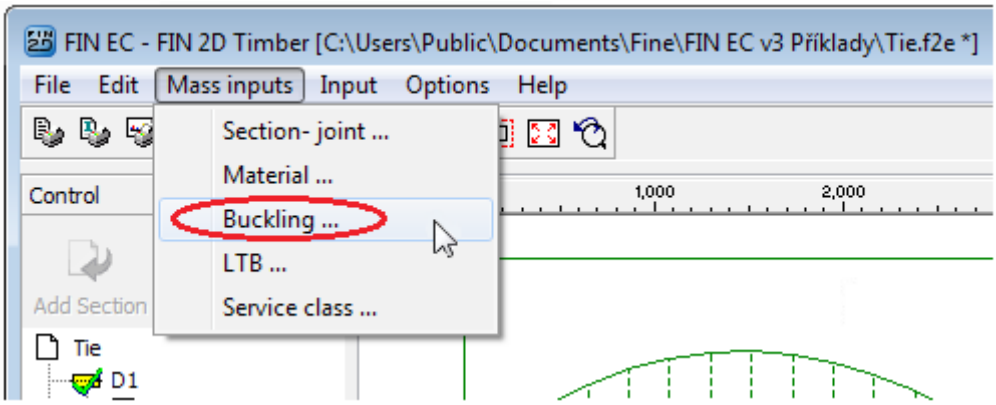
Editing geometry of the bottom chord

We carry out the design check again confirming more economic design of the member.



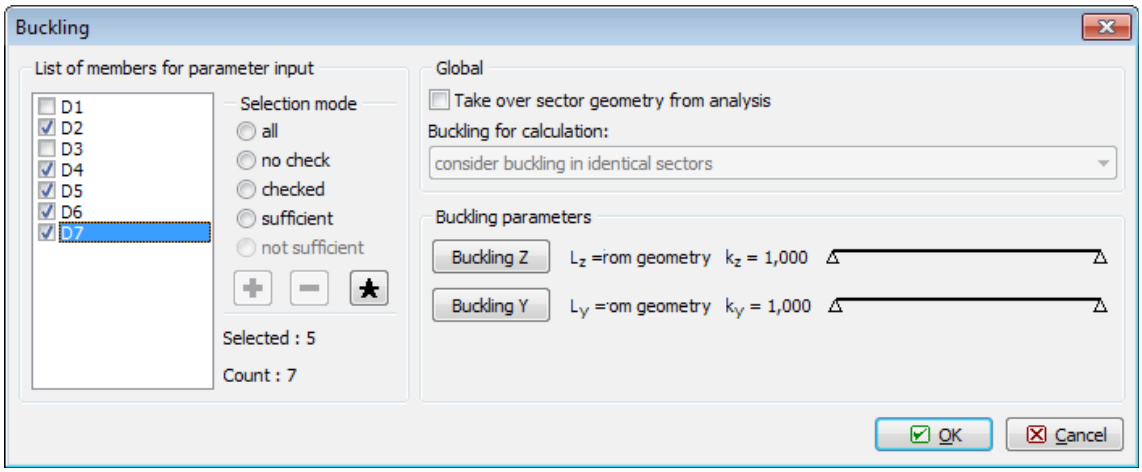
Bottom chord's design check

Finally, the diagonals remain to be checked. Because the diagonals' properties are almost identical, we can define the calculation parameters for all of them together. This can be done using a function in the "**Mass input**" part of the main menu. First we define the buckling parameters.



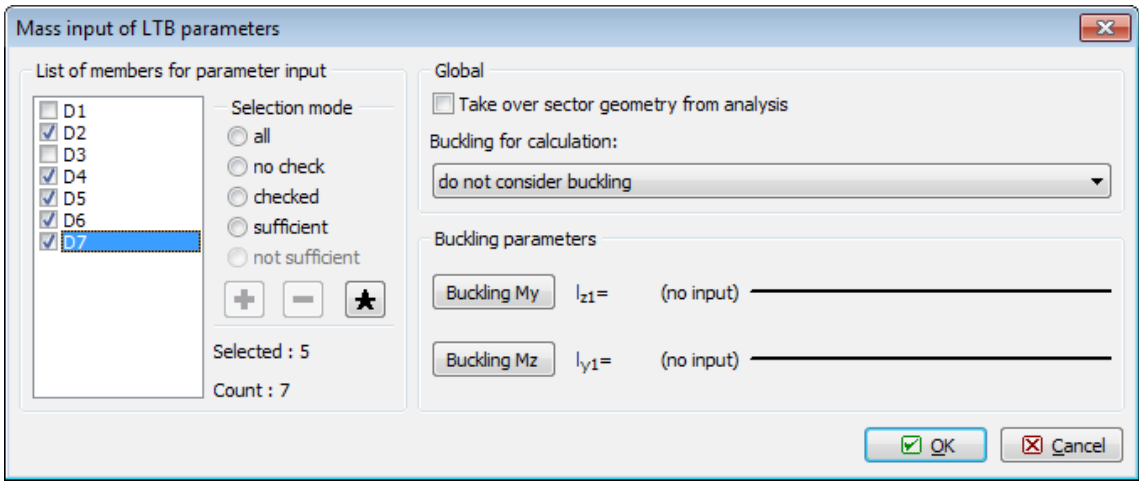
Mass input of buckling properties

Even though it would be sufficient to define the buckling parameters for the compressive members only, it is easier to assign them to all diagonals. Hence we select members D1 to D4 on the left-hand side. In the right part we tick the "**Take over sector geometry from analysis**" so that the program will use the actual lengths of the members as buckling lengths. Finally we define simple supports at both ends for both y and z directions. After clicking "**OK**" the entered parameters are assigned to all diagonals.



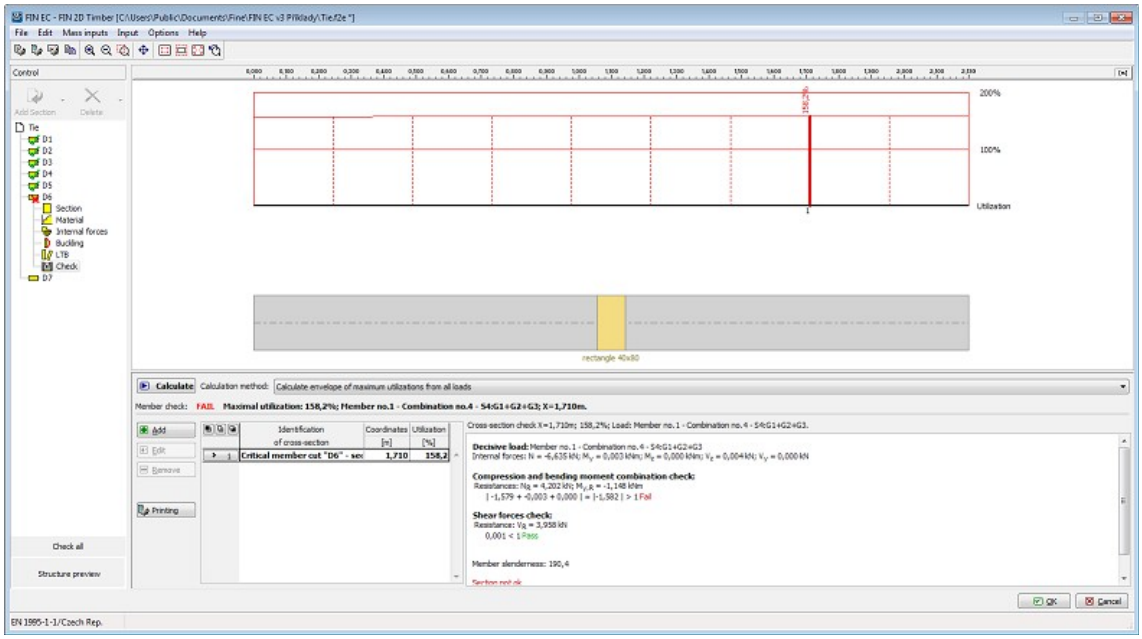
Mass input of buckling parameters

We continue with defining lateral torsional buckling parameters. Diagonals are generally only subject to axial forces therefore the LTB checks are not required; however, due to their self weight, small bending moments can occur. In such cases, the program demands lateral torsional buckling checks. In the "**Mass input**" section we select "**LTB**"; in the left part of the dialog box we again select all diagonals and from drop down list in the right part we select "**do not consider buckling**". Thus the influence of lateral torsional buckling will not be considered in the design checks of the selected members.



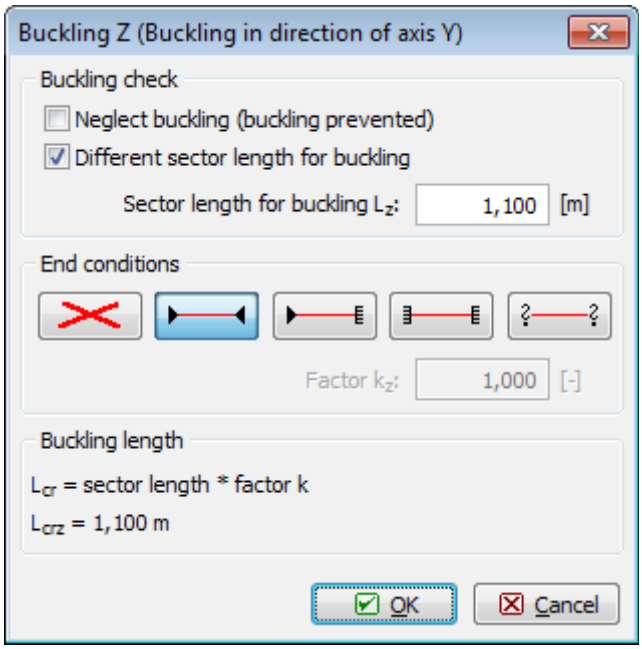
Mass input of buckling parameters

Now we can carry out the design checks for all diagonals. The diagonal D6 does not pass the buckling check, however we can increase its capacity by reducing its buckling length.



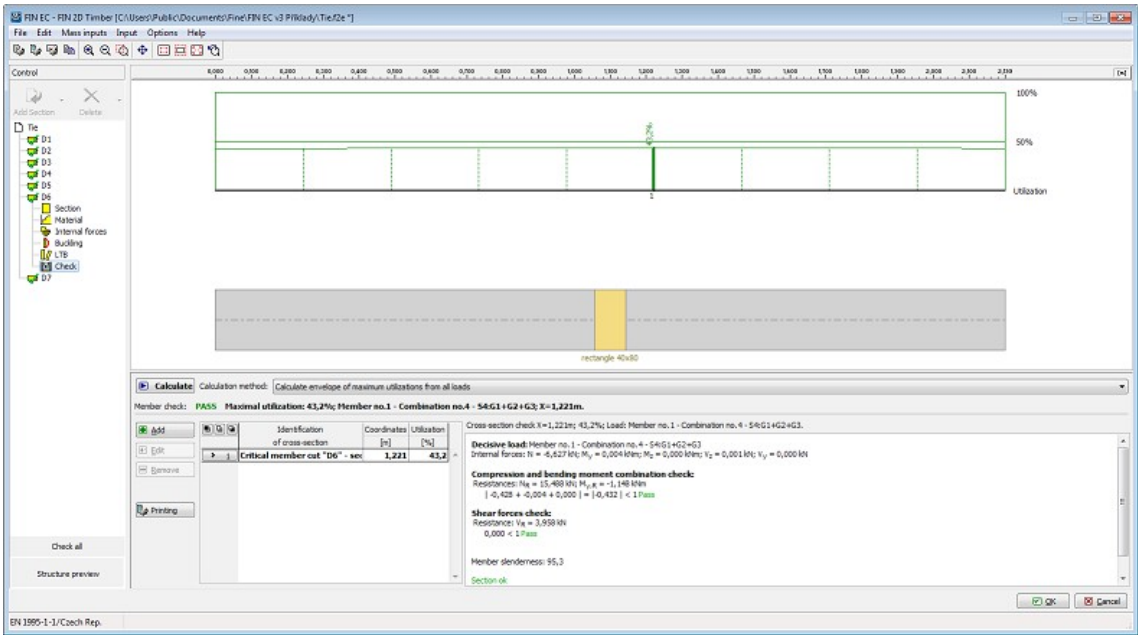
Diagonal D6 check

Therefore we design a longitudinal stiffener in the diagonal's centre which will reduce its buckling length to half. In the "**Buckling**" section of the control tree we adjust the buckling length in the "**Buckling Z**" dialog box.



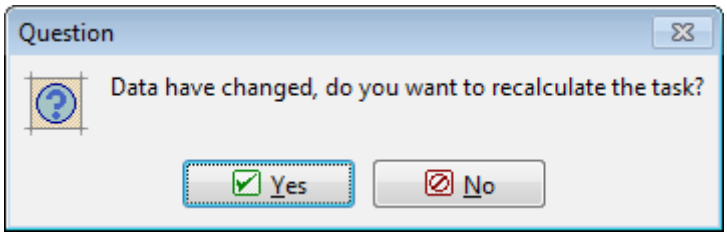
Adjusting buckling length

After re-calculating, the diagonal passes the design checks.



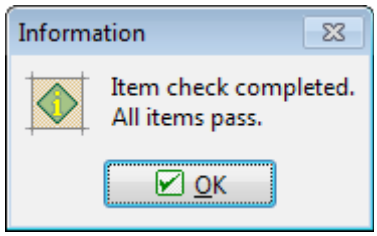
Checked design members

Now all design members and groups in the structure are checked, therefore we can exit the design module by clicking "**OK**". Program 2D has recognized that some members were changed. Because the stiffness of individual members changed and thus different internal forces can act on some of the them, it is necessary to re-calculate the structure. The program automatically offers this option. If we select "**No**", the program will erase results and return to the pre-processor. In our example, we select "**Yes**" after which the structure is re-calculated.



Question on re-calculating structure

After finishing the calculation, the program updates distribution of internal forces and deformations and erases the results in the design module. Therefore we run the design module again and make use of the command "**Check All**" located in the bottom part of the control tree. We do not have to re-define all parameters as they are saved after previous definition. The module will carry out design checks for all members and inform us about the results.



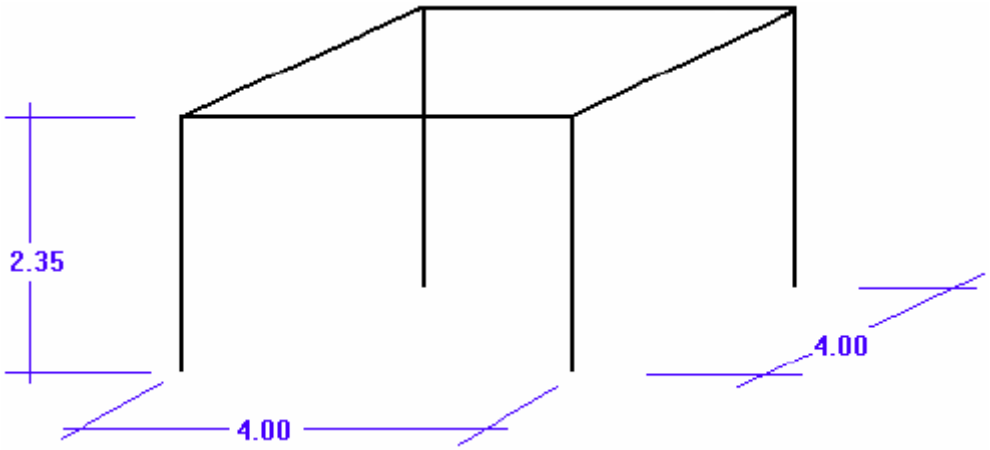
Members' design checks results

After returning to the program 2D, members which passed the checks are marked in green and the members which did not in red. In our case, all members are marked in green; therefore the structure's design is completed.

3D structure

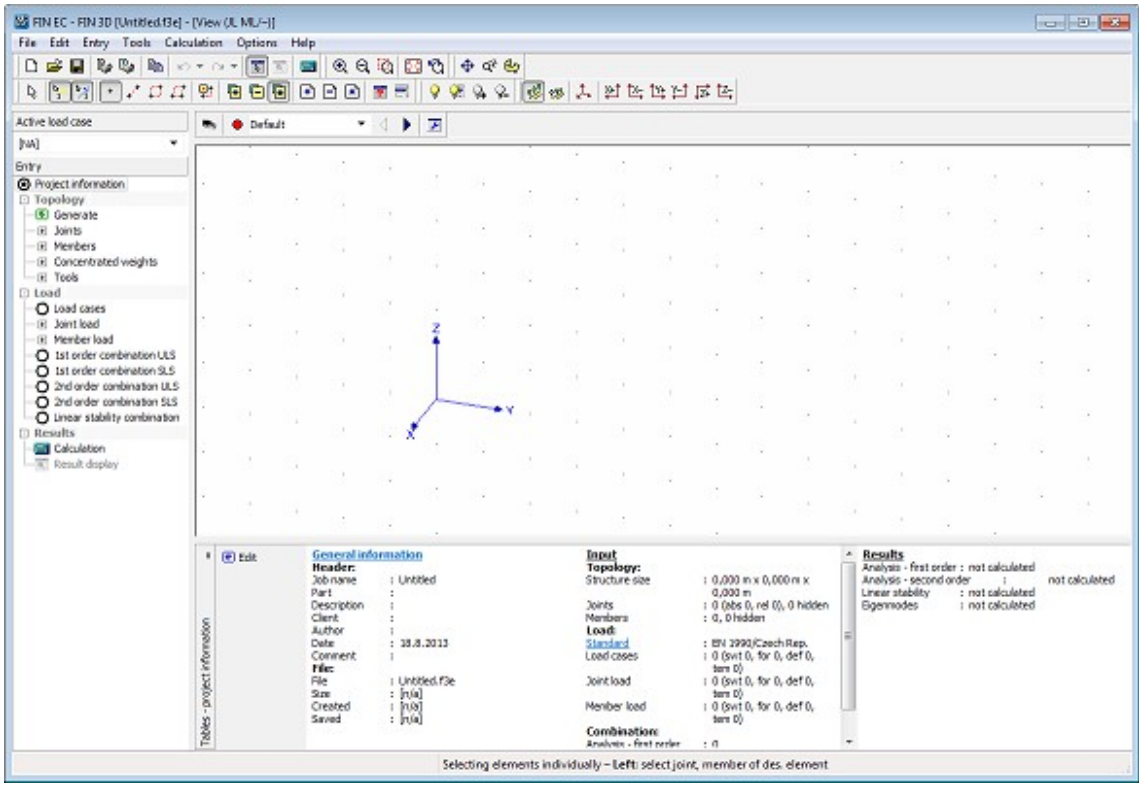
Task

In this example, the task is to design and check the 3D structure shown on the sketch below. The columns are circular hollow sections and the beams are *I*-sections, both made of steel of the grade *Fe 360*. The beams are subject to vertical uniformly distributed load *18 kN/m*; the lateral loading on 2 of the columns is trapezoidal with top magnitude *12 kN/m* and bottom magnitude *19 kN/m*. The columns are fixed to the concrete underlay.



Geometry of the structure

After running the program 3D, the following screen appears.



Program's main screen

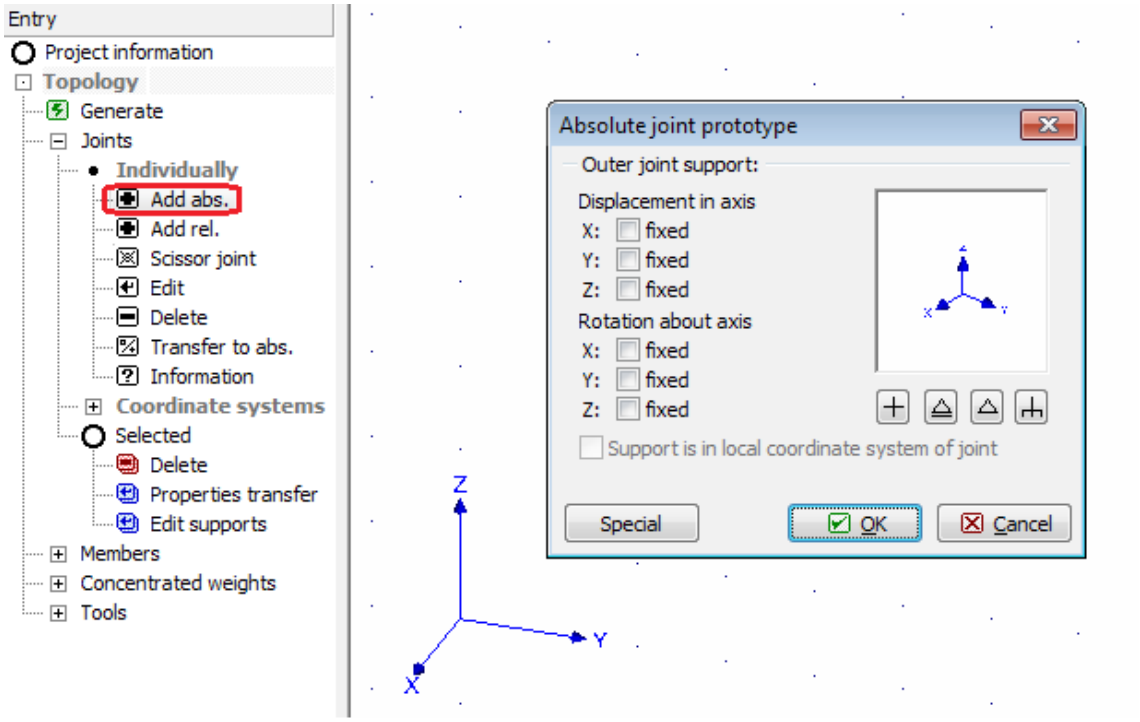
Structure's definition

In this example, we will not use the "**Generator of 3D structures**"; instead we will define the structure's geometry by entering individual joints and members. Thus we can show alternative

ways of creating and editing structures. Using this approach, there are more ways how to define the structure's geometry. In our example we opted for the following:

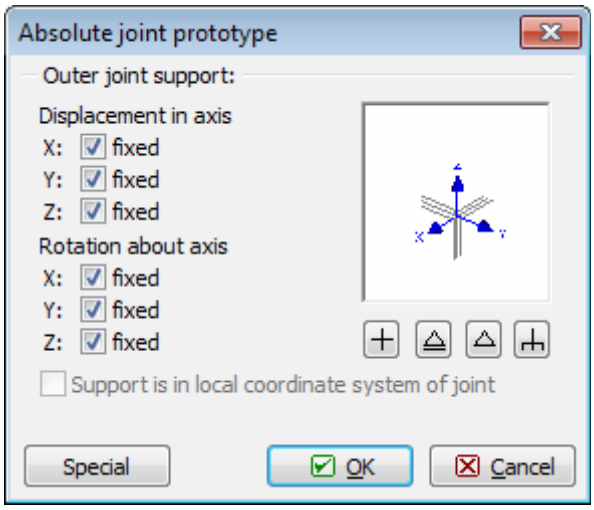
- first we define a frame (2 columns and a beam)
- we define the loading on members
- we copy the frame to obtain a 3D structure
- we define the transverse beams' properties including loading

This approach has not been selected because it would be ideal or simplest, but because it gives us the opportunity to explore more of the program's functionalities.
First we save the project, naming it e.g. "**3D structure**". The name is displayed in the main screen's header and it will also be exported to the design programs.
In the control tree in the left part of the screen, we select "**Joints**" and "**Add abs.**". A dialog box for joint prototype's definition appears; its purpose will be further explained.



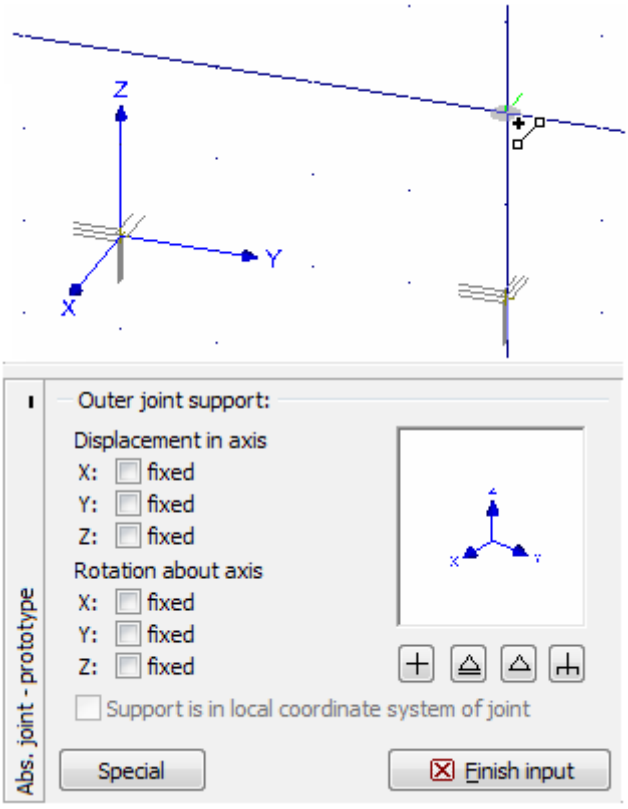
Absolute joint prototype definition

The purpose of the dialog box "**Absolute joint prototype**" is to define the fixed directions of the entered joints. First we will define the joints of the fixed column bases either by ticking the "**fixed**" boxes for all directions, or by clicking the button with a symbol of a fixed base on the right-hand side of the dialog box. We confirm the support definition by clicking "**OK**".



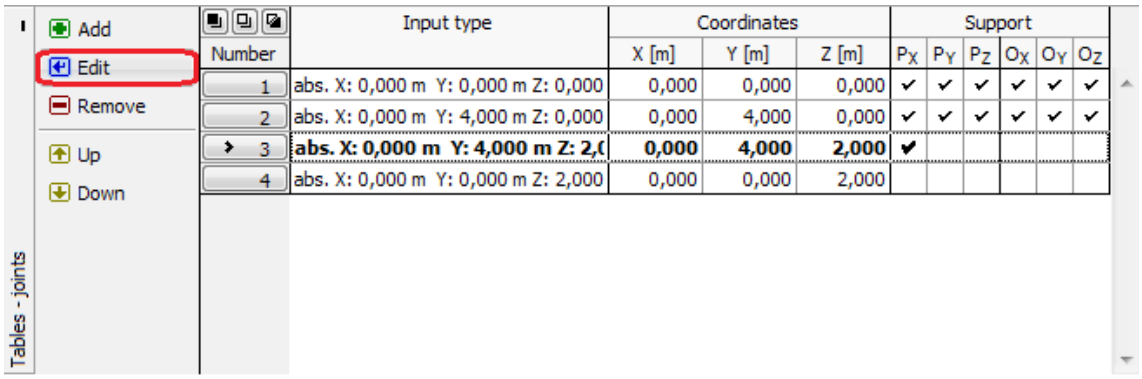
Absolute joint prototype defined as a fixed base

The dialog box "**Absolute joint prototype**" moves into the bottom definition frame. Now we can enter nodes with coordinates *[0,0,0]* and *[4,0,0]* in the model space; for easier entering we make use of the default grid step of *1 m*. We can check the coordinates in the status line of the main screen. During the nodes definition, the information about the joint's prototype remains in the definition frame, thus we can always control which joint (or member or loading) is being defined and we can easily change the prototype's parameters to continue with definition.
For upper nodes' definition, we need to change the prototype's parameters; as they will not be supported, we untick the fixity in all directions (or we use the button with a symbol of a simple node on the right-hand side). Then we enter the nodes with coordinates *[0,0,2]* and *[4,0,2]*.



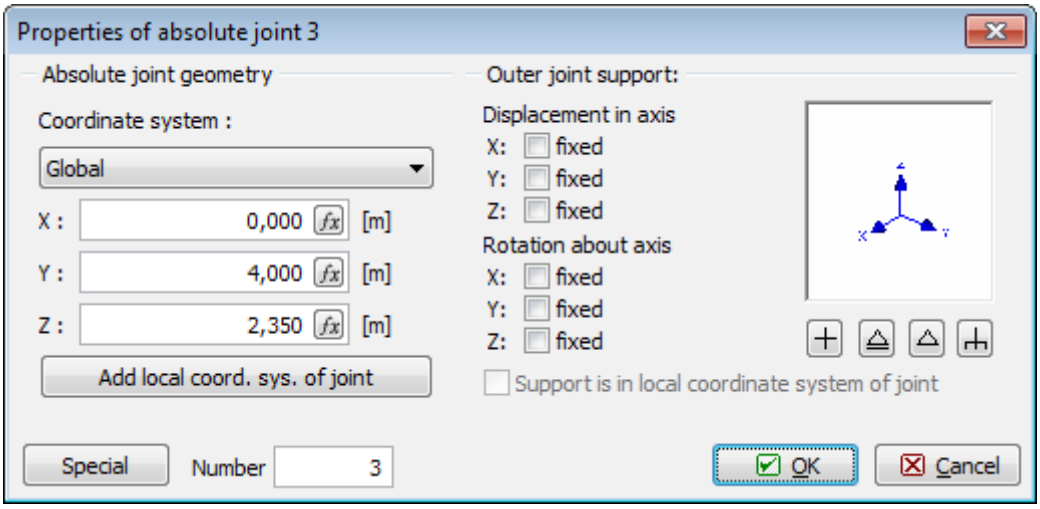
Definition of the upper nodes

Because the upper joints are now aligned to the grid step, their coordinates in the z-direction are *2.0 m* instead of *2.35 m* as given in our example. We therefore edit the coordinates of these joints by selecting them in the table located below the model space (the active joint is highlighted in bold) and click the "**Edit**" button.


Add
Edit
Remove
Up
Down
Tables - joints
Number
Input type
Coordinates
X [m]
Y [m]
Z [m]
Support
PX PY PZ OX OY OZ
1 abs. X: 0,000 m Y: 0,000 m Z: 0,000 0,000 0,000 0,000
2 abs. X: 0,000 m Y: 4,000 m Z: 0,000 0,000 4,000 0,000
3 abs. X: 0,000 m Y: 4,000 m Z: 2,(0,000 4,000 2,000
4 abs. X: 0,000 m Y: 0,000 m Z: 2,000 0,000 0,000 2,000

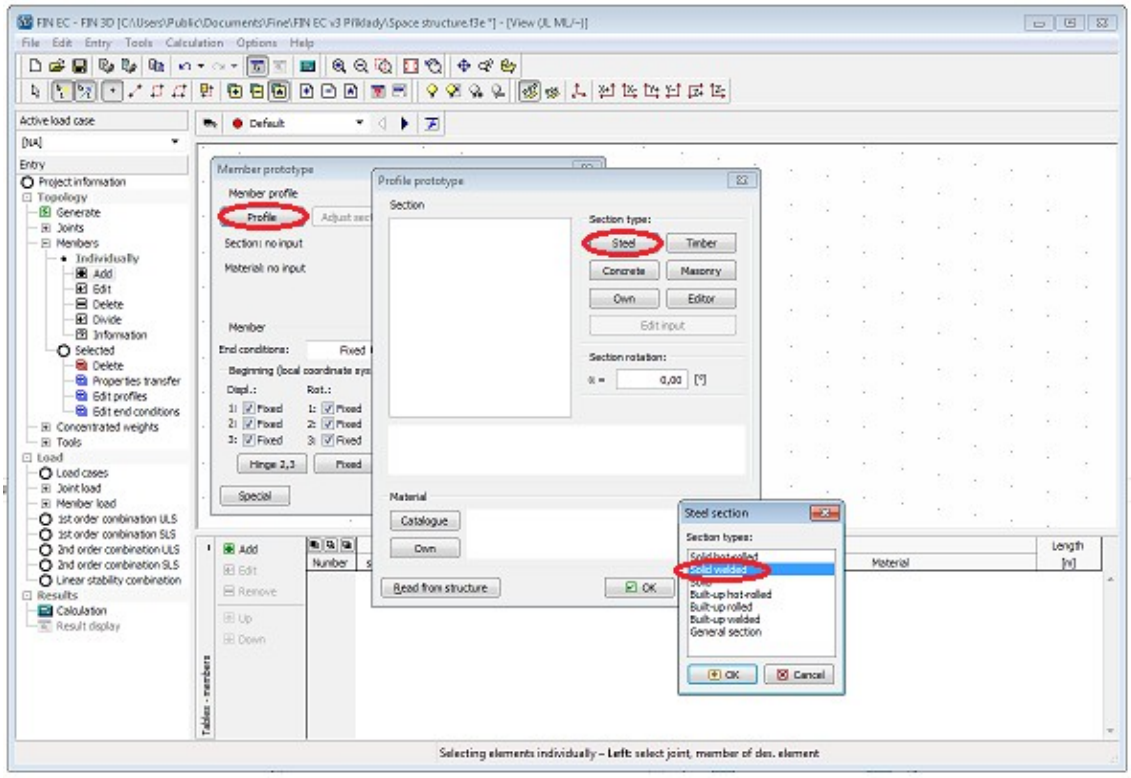
Editing joints

In the dialog box "**Properties of absolute joint**" we change the Z coordinate to *2.35 m*. We confirm the definition by clicking "**OK**" and edit the fourth joint analogically.



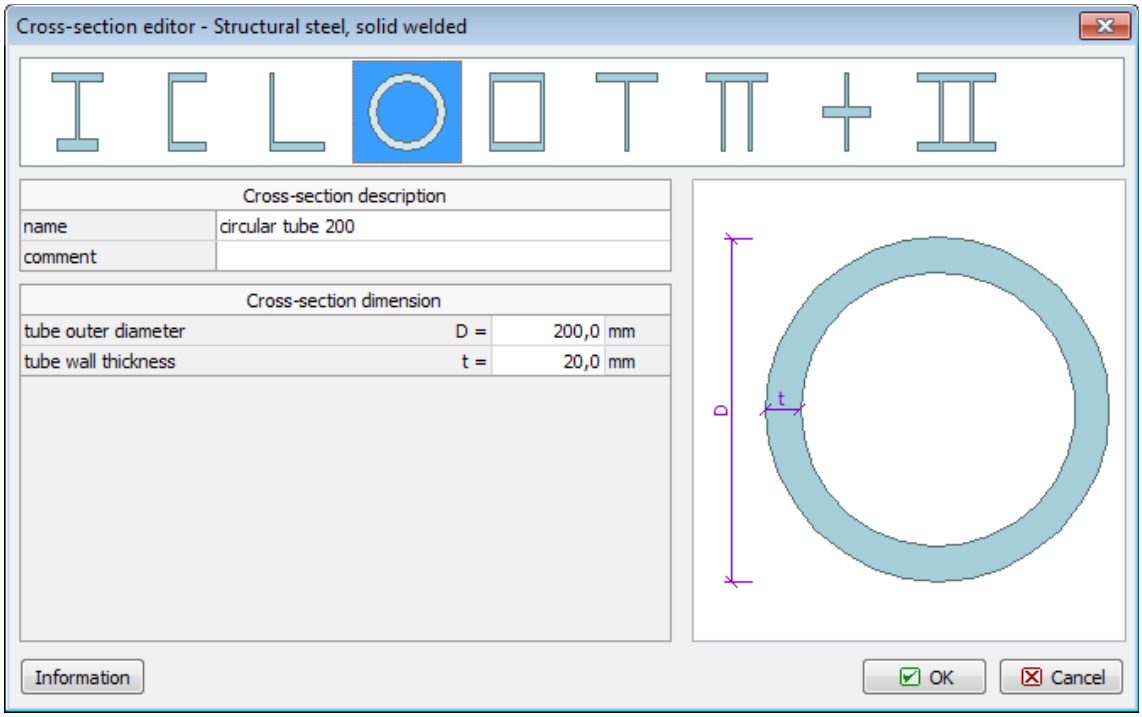
Change of absolute joint's coordinate

Now we can define the members in the same way as the joints. We switch to "**Members**" and "**Add**" in the control tree; similarly to joints definition, a dialog box with member prototypes' properties appears. In this dialog box, we define the cross-section using the "Profile" button and selecting the section type "**Steel**" and "**Solid welded**" in the following dialog boxes.



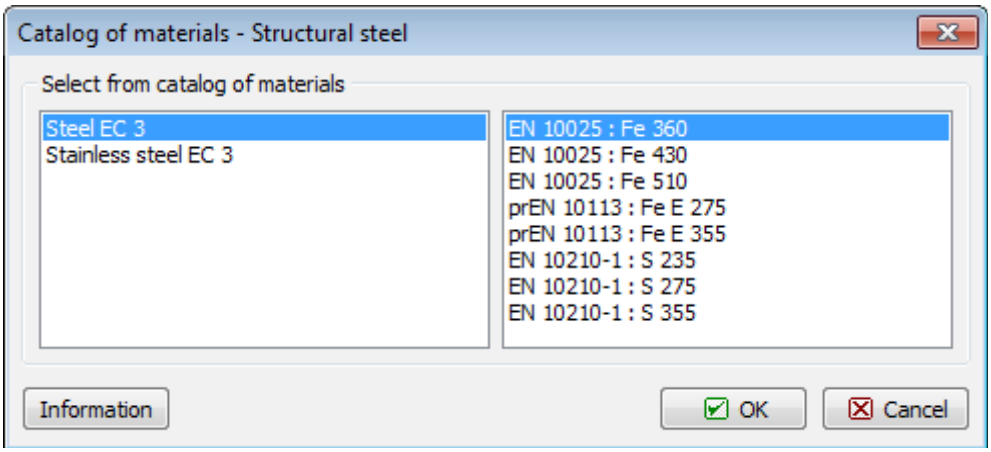
Selection of the section type

In the "**Cross-section editor**", we select a circular tube and define the *200 mm* diameter and *20 mm* wall thickness.



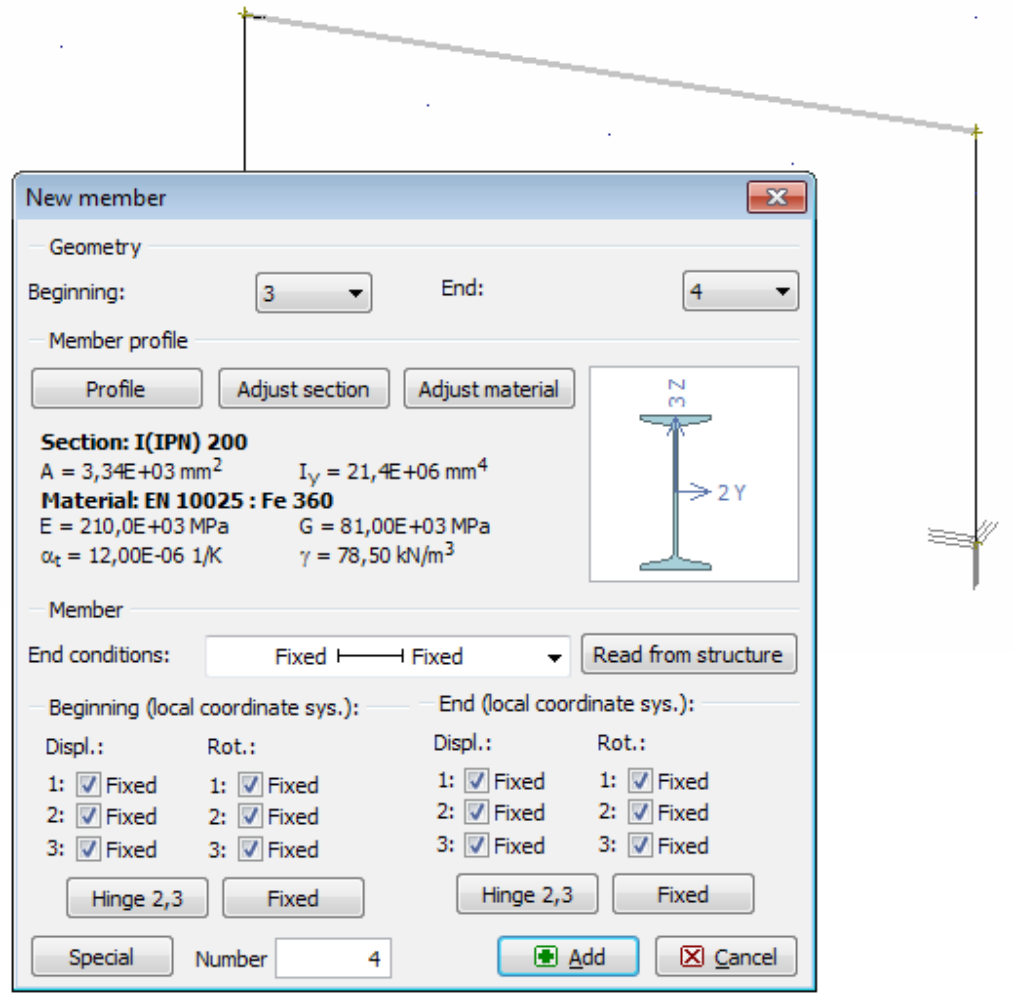
Defining cross-section dimensions

In the following dialog box, we select the steel grade *Fe 360*.


Catalog of materials - Structural steel
Select from catalog of materials
Steel EC 3
Stainless steel EC 3
EN 10025 : Fe 360
EN 10025 : Fe 430
EN 10025 : Fe 510
prEN 10113 : Fe E 275
prEN 10113 : Fe E 355
EN 10210-1 : S 235
EN 10210-1 : S 275
EN 10210-1 : S 355
Information
OK
Cancel

Selecting steel grade

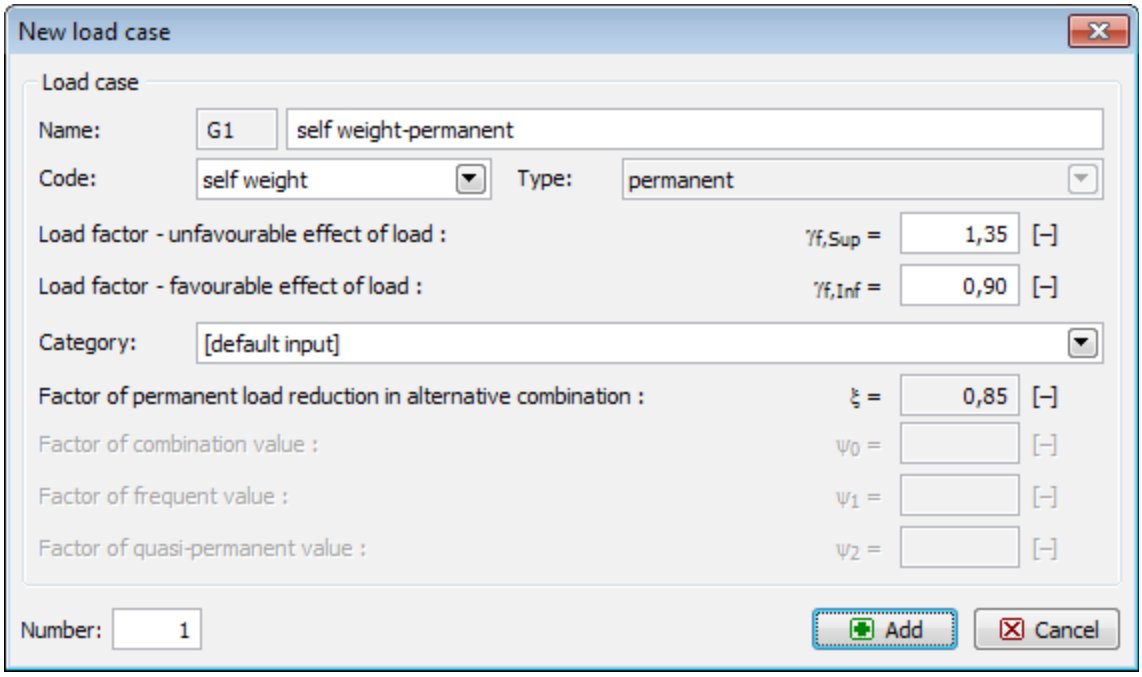
To define the member's prototype, we click on the beginning and then the end joints; this way we define both columns. During the definition we try to keep the same orientation of the members, hence in both cases we first select the lower joints and then the upper (or vice versa). The columns will thus have the same local direction which will later simplify their design.
Before defining the beam, we change the prototype's cross-section to *I200*. Then we connect the tops of both columns with the beam or define the beginning and the end joints again.



Beam definition

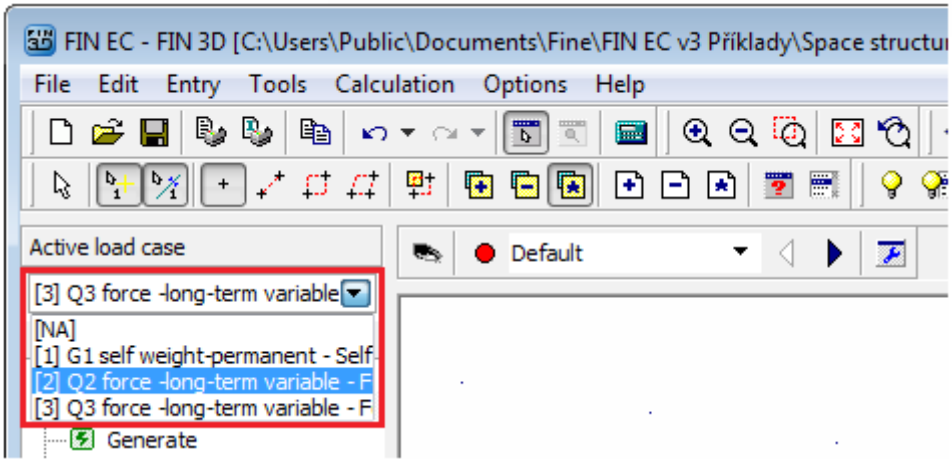
Loads definition

Now we proceed to defining the loading. We switch to "**Load cases**" in the control tree and using the "**Add**" button by the definition table we run the dialog box "**New load case**". The program automatically offers the first load case "**self weight – permanent**". This load case is specific as it contains only automatically generated loads from the structure's self weight which are immediately updated when any change of a member or a material is made. This load case can be defined only once. Definition of the load case "**self weight – permanent**" is confirmed by the "**Add**" button.



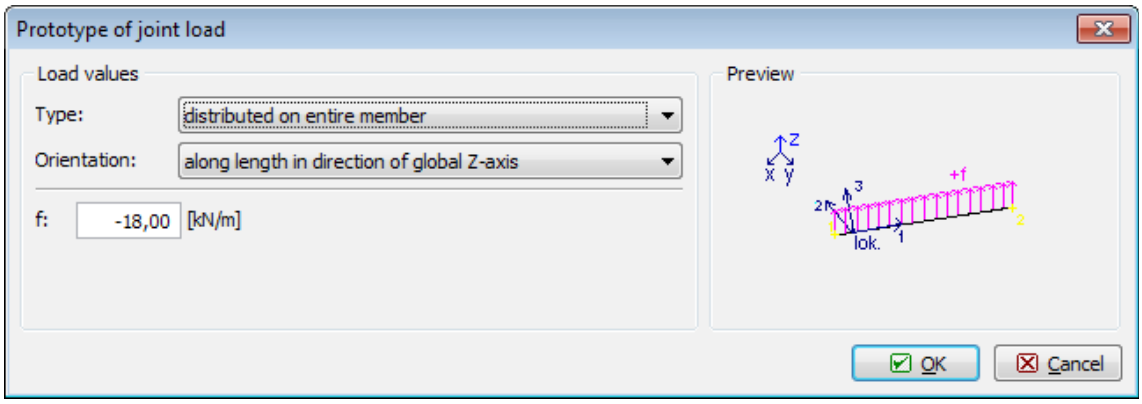
"Self weight" load case properties

Analogically we add another two load cases for which we select the type "**long-term variable**". Now we need to input the loads for each of the defined load cases. We switch to "**Member load**" in the control tree and select the load case 2 as the active one in the top left corner.



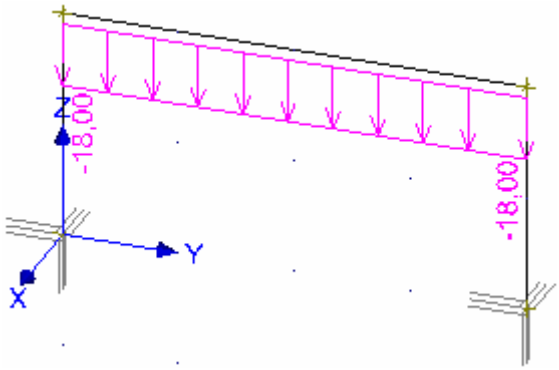
Selecting an active load case

We switch to "**Member load**" and "**Add**" in the control tree. In the "prototype of joint load" dialog box we select the load type "distributed on entire member" and the load orientation "along length in direction of global *Z-axis"*, and enter the value *-18 kN/m*.



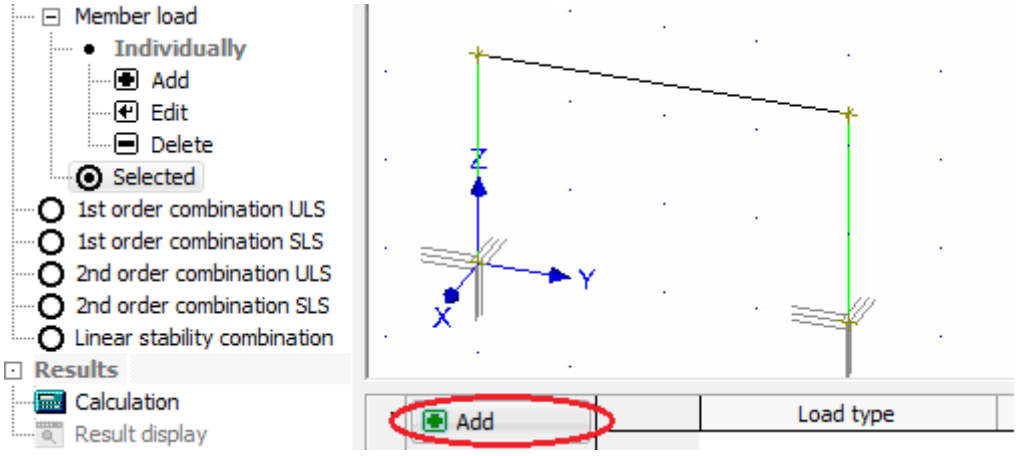
Prototype of joint load

We apply the load by clicking on the beam in the model space; the load is instantly displayed on the structure.



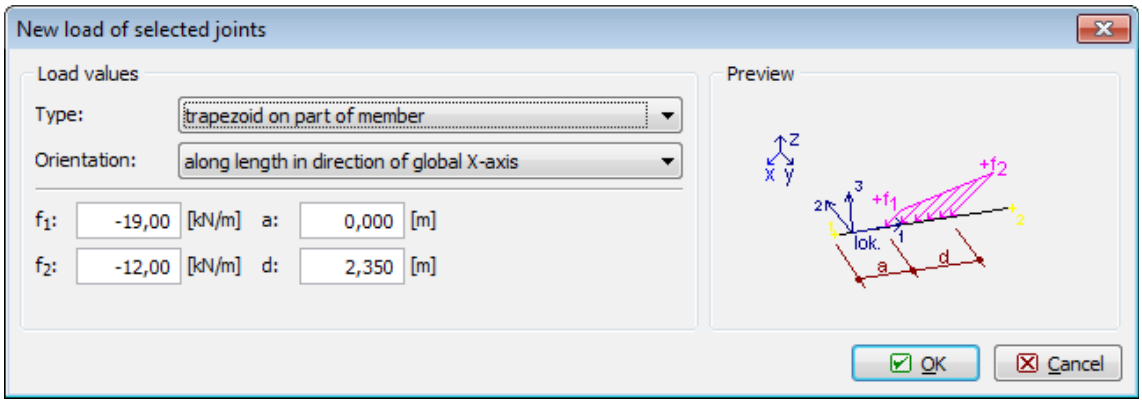
Structure with defined load

We change the active load case in the drop down list in the top left corner to the load case 3. In this load case, we input the trapezoidal loads on columns. As the loads are the same for both columns we can use collective definition for selected members. We switch in the control tree to "**Member load**" and "**Selected**". First we select both columns by clicking on them in the model space; they become highlighted in green. Then we click the "**Add**" button by the definition table.



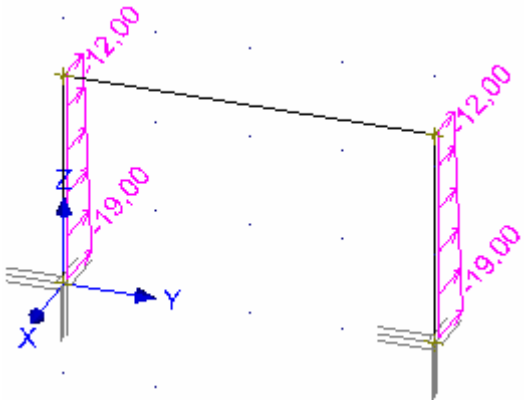
Button for load definition for selected members

In the dialog box "**New load of selected joints**" we select the type "trapezoid on part of member" and the orientation "along length in direction of global X-axis". Subsequently we enter the magnitudes at the beginning and at the end as well as the segment length.



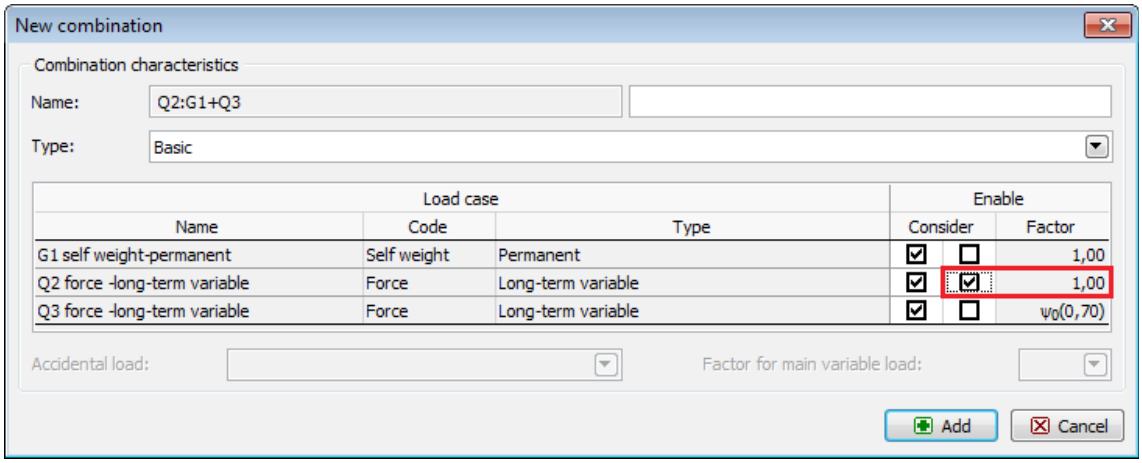
The dialog box for load definition for selected members

After clicking "**OK**", the loads are applied on both selected members. By repeated clicking on the members we deactivate their selection.



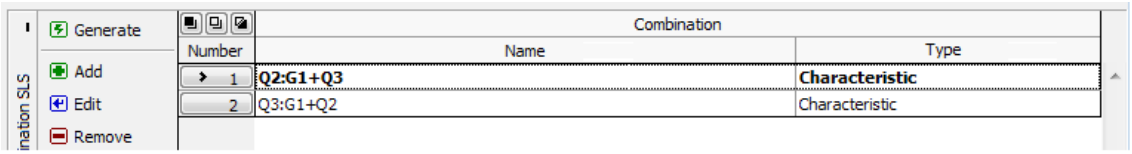
Structure with defined trapezoidal loads

Definition of load combinations is the last step in the loading definition. In the program Fin 3D, the combinations for ultimate and serviceability limit states are distinguished. First we create the ultimate limit state load combination. Only two combinations will suffice for the design. Both will contain all load cases; in the first, the load case 2 will be considered main variable whereas the load case 3 will be considered the main variable in the second. For definition we use the "**Add**" button after switching to "**1st order combination ULS**" in the control tree. In the table located in the lower part of the dialog box "**New combination**" we select all load cases by ticking the boxes in the first column of the "**Consider**" section. In the second column, we tick the box only for the load case 2. This way we set this load case the main variable. We confirm the definition of the combination by clicking the "**Add**" button.



Selection of the main variable load case

Then we change the setting so that the load case 3 is considered the main variable and again click the "**Add**" button. Analogically we define the two "**Characteristic**" combinations in the "**1st order combination SLS**" section of the control tree.

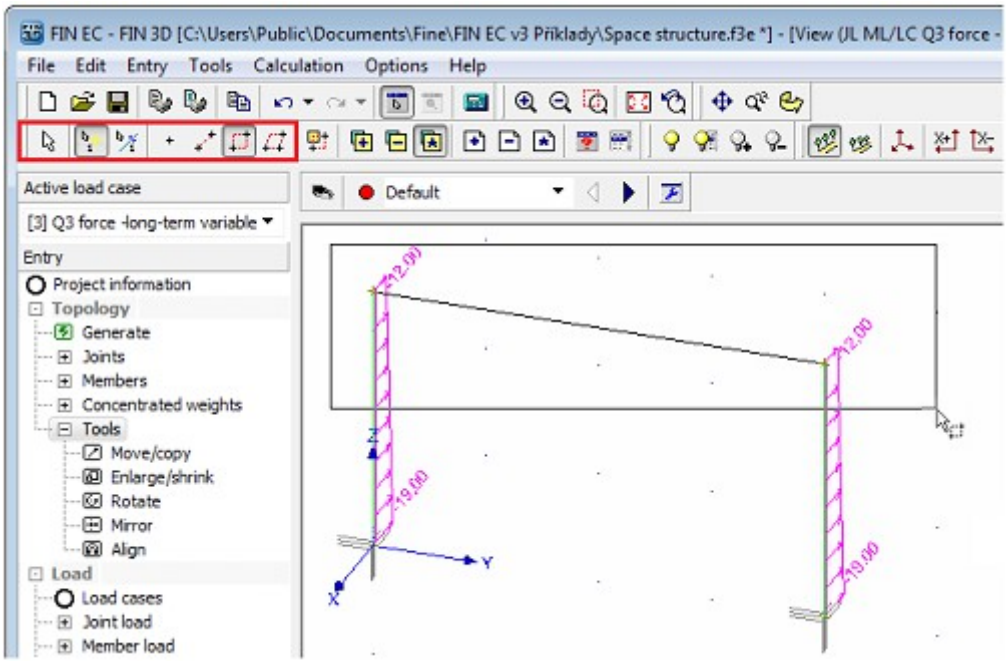


List of combinations for serviceability limit state

Thus we finished the definition of loading and we can proceed to expanding the structure to 3D.

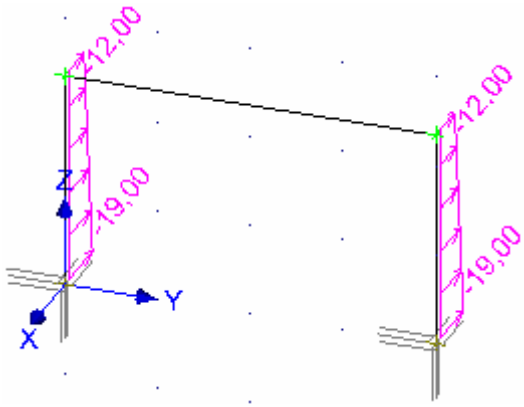
Editing structure

In this chapter, we will expand the structure to a 3D frame. We switch to the section "**Topology**" and "**Tools**" of the control tree. First we need to select the upper joints because during copying, the program is able to create in these locations the new beams connecting the original 2D frame with the copied one. To be able to select the joints easily, we edit the selection setting in the "**Selection**" toolbar. Firstly, we switch off the "**Member selection**" so that the program can only select the joints; secondly we change the selection mode to rectangular. Then we can easily select the upper joints.



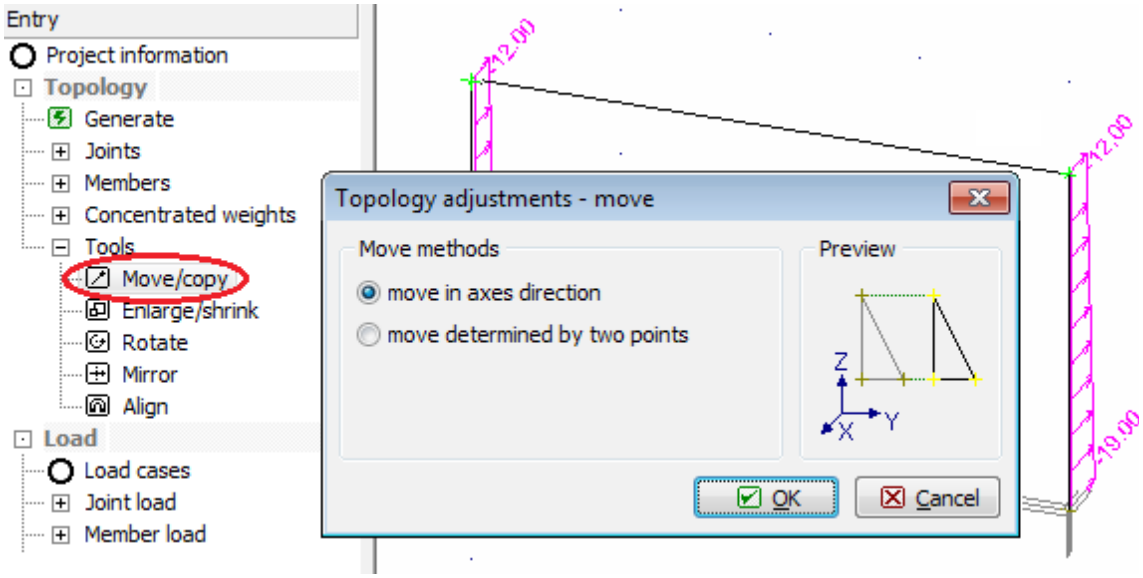
Selection mode tools

The selected joints are highlighted in green.



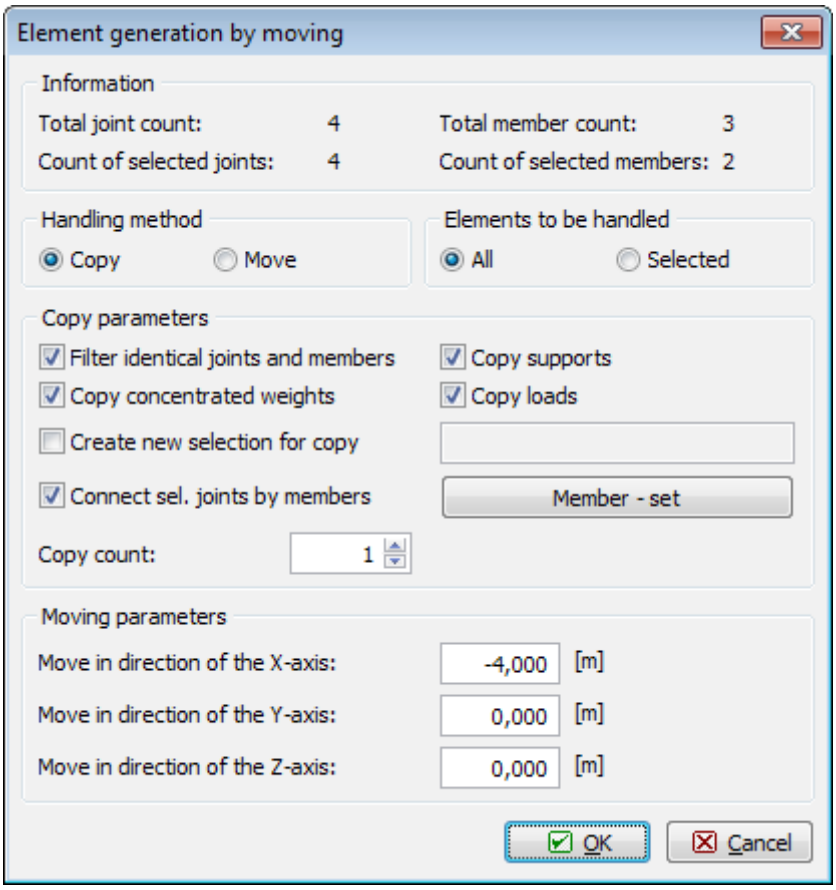
Structure with selected nodes

Now we select "**Move/Copy**" in the control tree. A dialog box appears in which we can select whether we will move the structure in the direction of the global axes or in a direction defined by two nodes. We select the first option and confirm by clicking "**OK**".



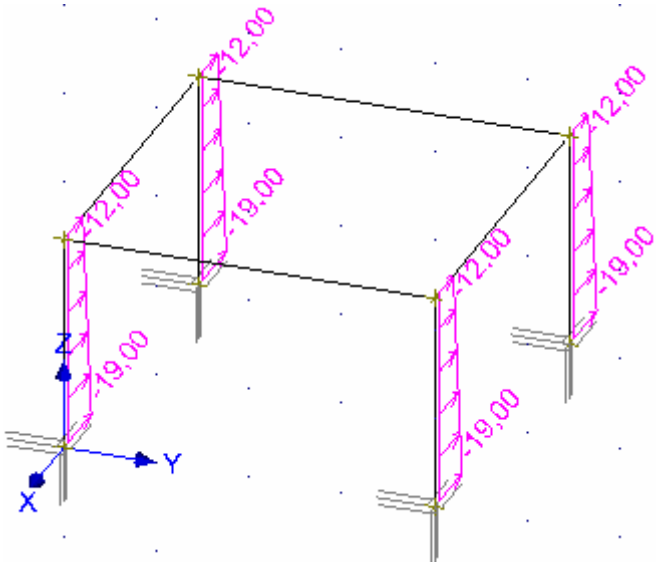
Move/copy tool

In the dialog box "**Element generation by moving**" we set the necessary parameters for moving the structure. We select the handling method "**Copy**" and enter the value of the "Move in direction of the X-axis" *-4 m*. Also we tick the box "**Connect sel. joints by members**", as result of which the members connecting the original frame with its copy will be inserted. Clicking the button "**Member – set**" we can confirm that the inserted members are of the section *I200*.



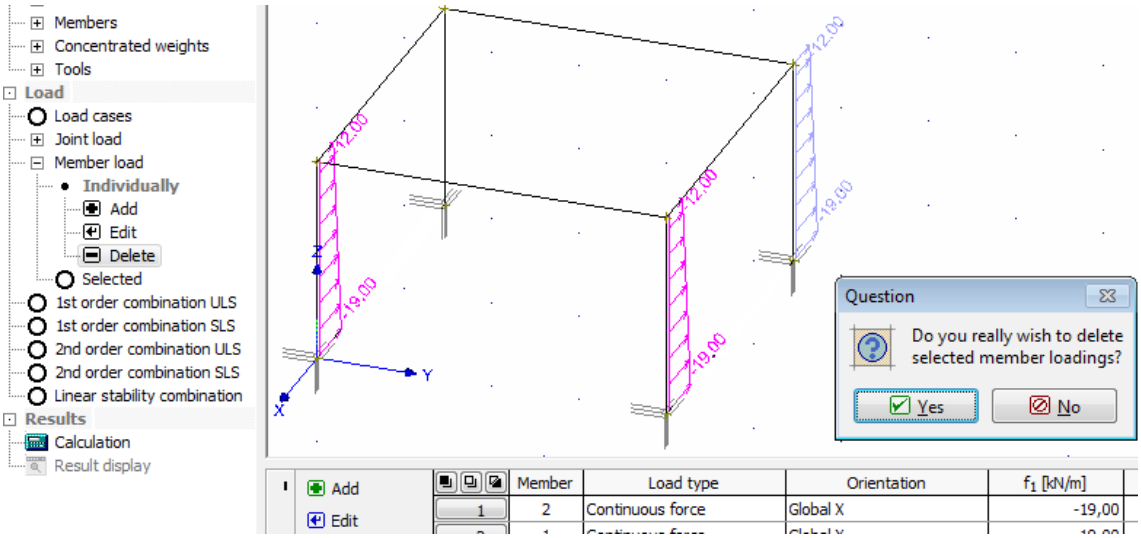
Moving parameters

After clicking "**OK**", the 3D structure is created. Using the button in the "**Selection**" toolbar we deactivate selection of the joints.



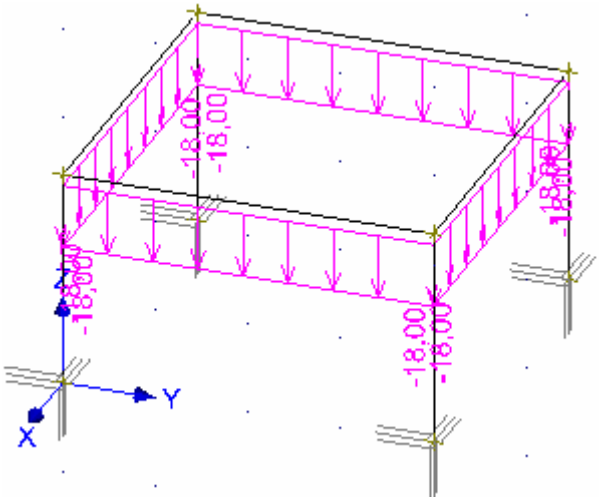
Created 3D structure

Now we need to edit loading for individual load cases. In the active load case 3 there is now the trapezoidal loading applied to all columns. However, in accordance with our task we want to apply the loading to the front columns only. Therefore we switch to "**Member load**" in the control tree and remove or delete the loads from the newly created columns.



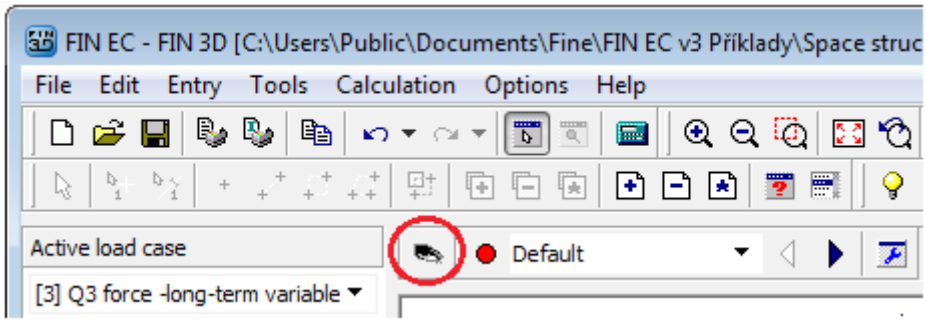
Removing loads from members

We also edit the loads in the load case 2. To input additional loads to this load case we first need to select the load case in the drop down list in the top left corner. Then we add the loads on the new beams with the same magnitude as for the other beams.



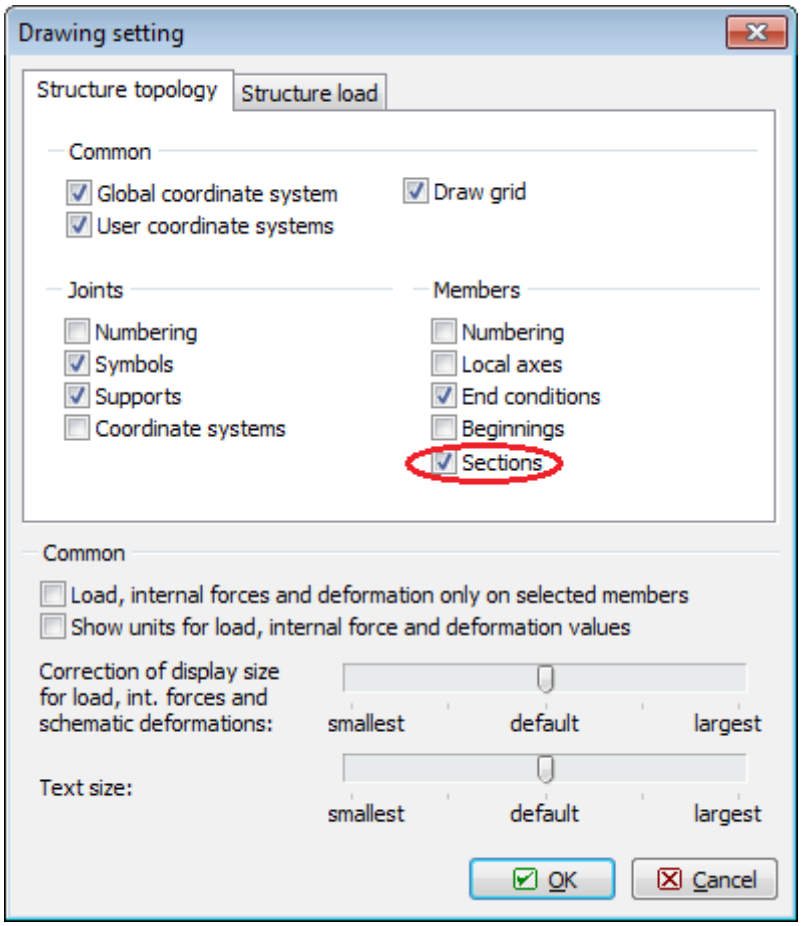
Loads in load case 2

The program offers a number of different ways and styles of displaying the structure. The dialog box for drawing settings is run by clicking the left button in the relevant view's toolbar.



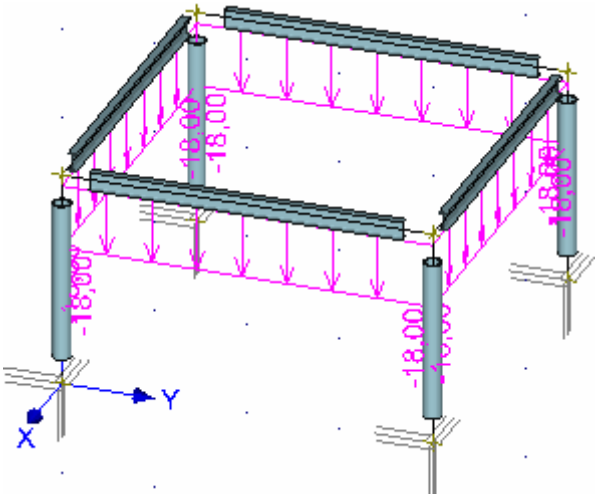
Button for setting the display parameters

In this dialog box we can for instance set displaying of members' sections.



"Drawing setting" dialog box.

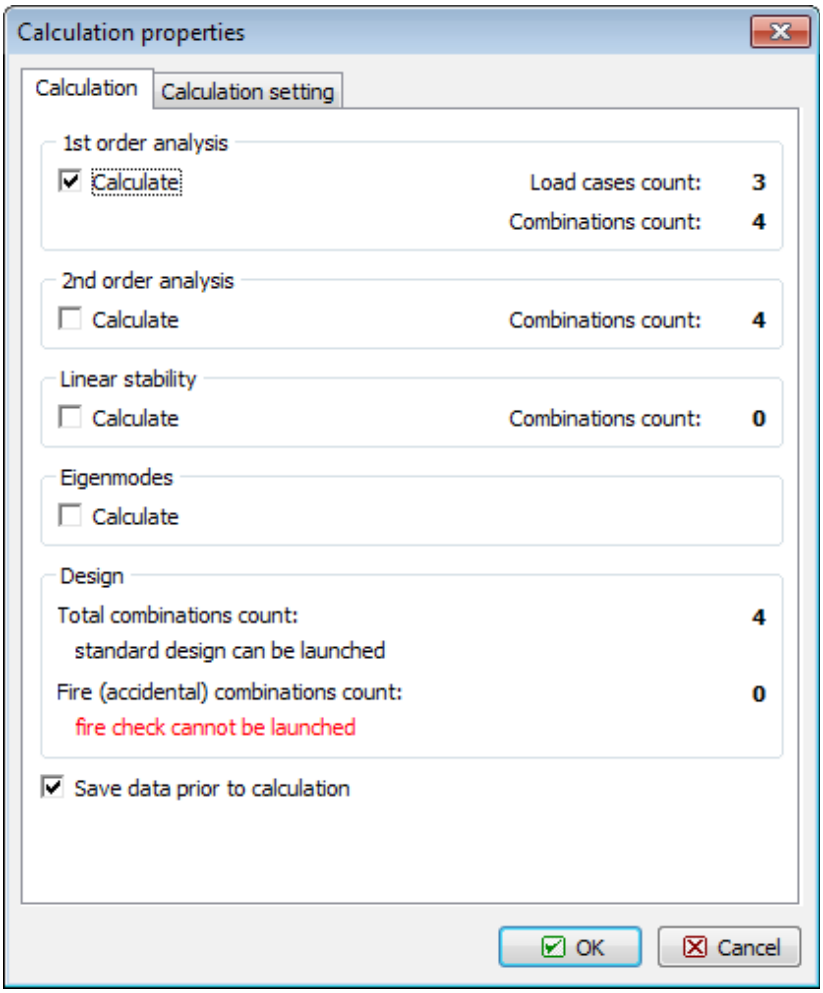
After closing the dialog box "**Drawing setting**", the members are displayed in 3D instead of being displayed as simple lines.



Structure with displayed sections

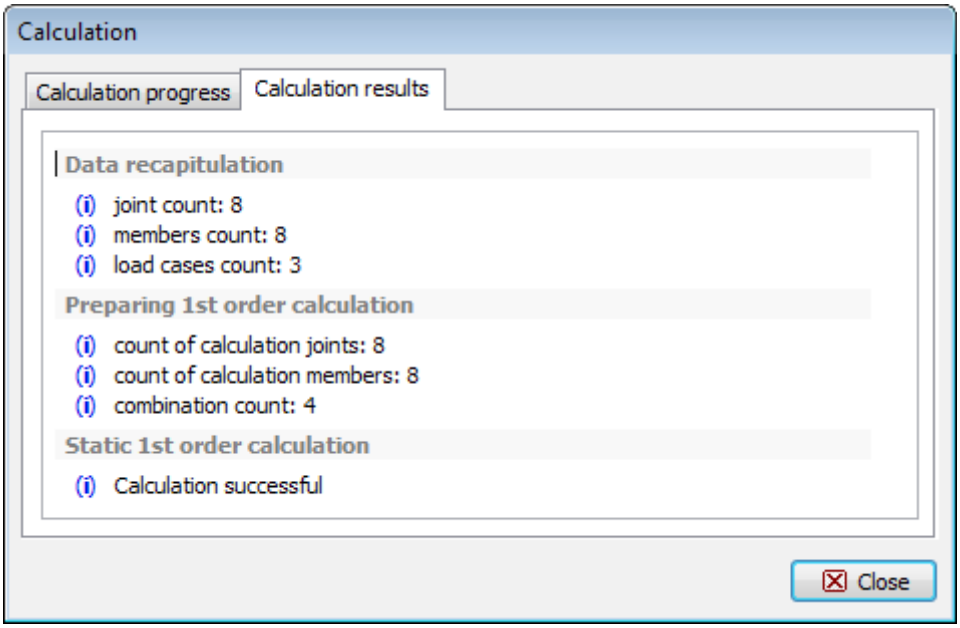
Calculations and results

Now we are ready to start the calculation. It is run by selecting "**Calculation**" in the "**Results**" section of the control tree. First the dialog box "**Calculation properties**" is displayed, and after confirming the settings, calculation is executed.



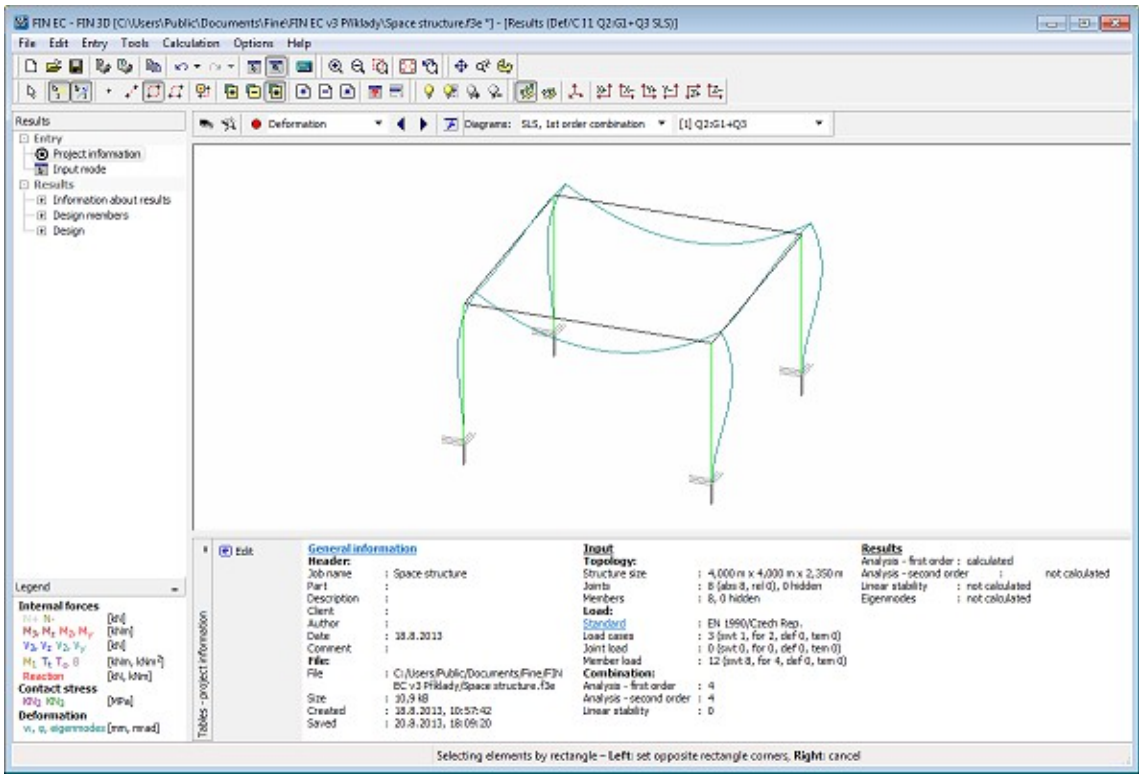
"Calculation properties" dialog box

The program executes the calculation; information about the progress and the results are displayed in the "**Calculation**" window, which we exit by clicking the "**Close**" button.



"Calculation" window with information about the calculation progress

After finishing the calculation, the program automatically switches to the post-processor and displays the deformed structure.

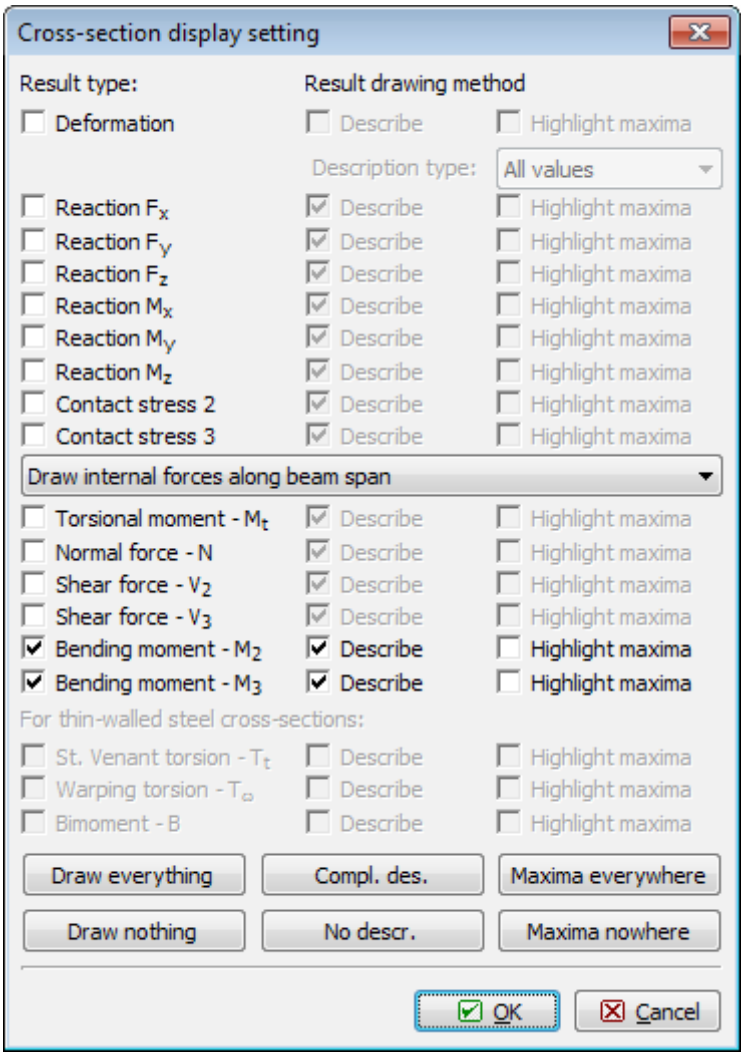


Deformed structure preview

The program contains a wide variety of tools for displaying individual diagrams. Apart from the fore-mentioned "**Drawing setting**", we can above all make use of the dialog box "**Cross-section display setting**" which is run by a button in the toolbar of each view. Because this setting is selected for each view separately, we can display different diagrams in different views.

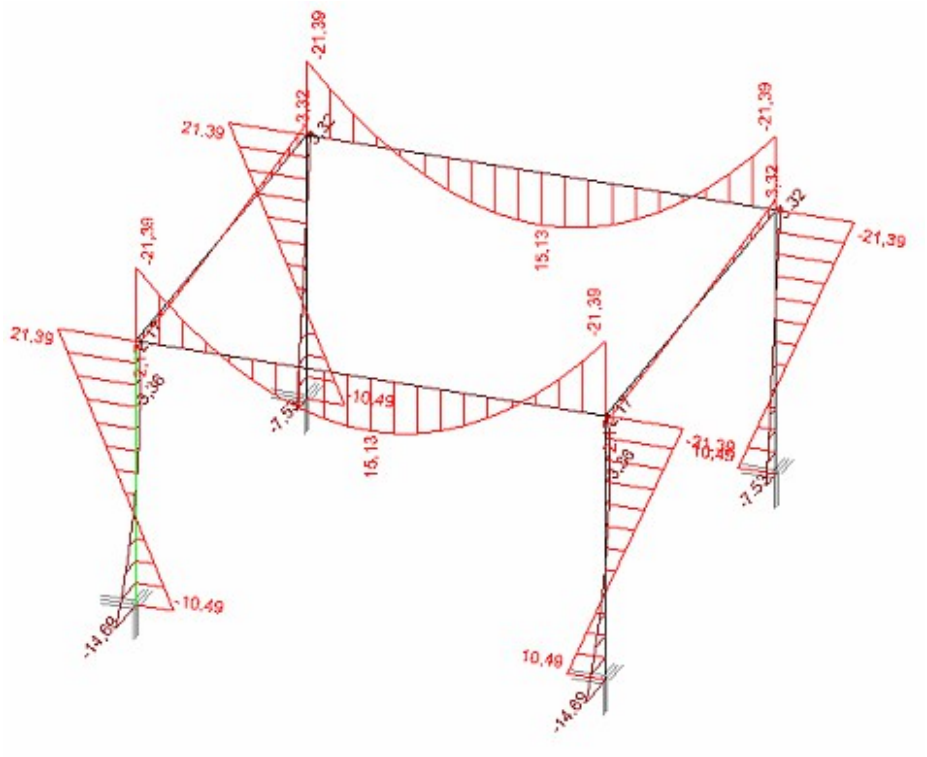
Button for running the "Cross-section display setting" dialog box

In this dialog box we can set which diagrams are to be displayed on the structure. For instance we can select bending moments.



"Cross-section" display setting dialog box.

As a result of changes made in the dialog box, the bending moment diagrams are now displayed on the structure.

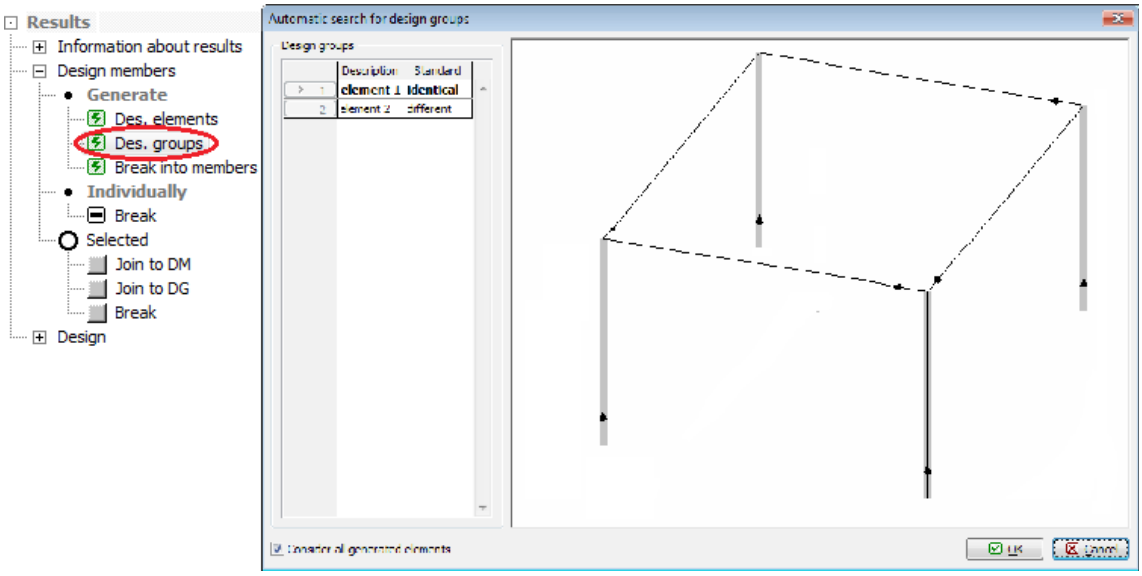


Bending moment diagrams on the structure

We can save the individual views and settings in the 'views manager' (run also from the toolbar of each view) and load them again anytime.

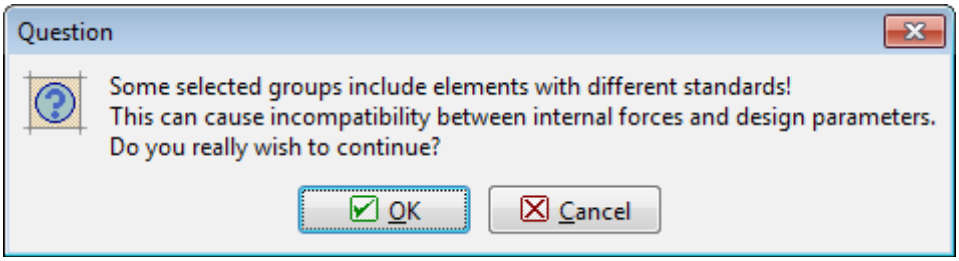
Design checks

In the next step we carry out the design checks of the structural members. In order to avoid checking each of the eight members separately, we first merge them into two design groups – columns and beams. A design groups acts as a single member which is checked for design forces in each of the members included in the group. The easiest way how to create a design group is to run automatic generation by selecting "**Results**", "**Generate**" and "**Des. groups**" in the control tree. In the dialog box "**Automatic search for design groups**", the programs offers two design groups (columns and beams). In the right part of the window a preview is displayed, in which we can check the suggested design groups.



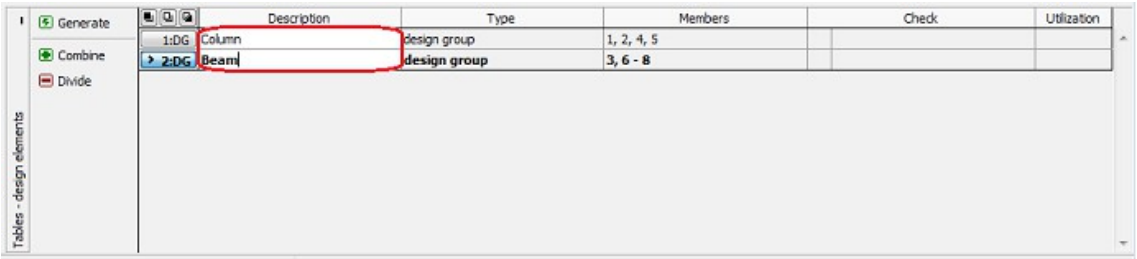
Automatic generation of design groups

After clicking "**OK**", the program notifies us that the members in the beams' design group have different orientation. Because we are not going to set unsymmetrical flexural nor lateral torsional buckling parameters to these members, we do not have to consider this notification.


Question
Some selected groups include elements with different standards!
This can cause incompatibility between internal forces and design parameters.
Do you really wish to continue?
OK
Cancel

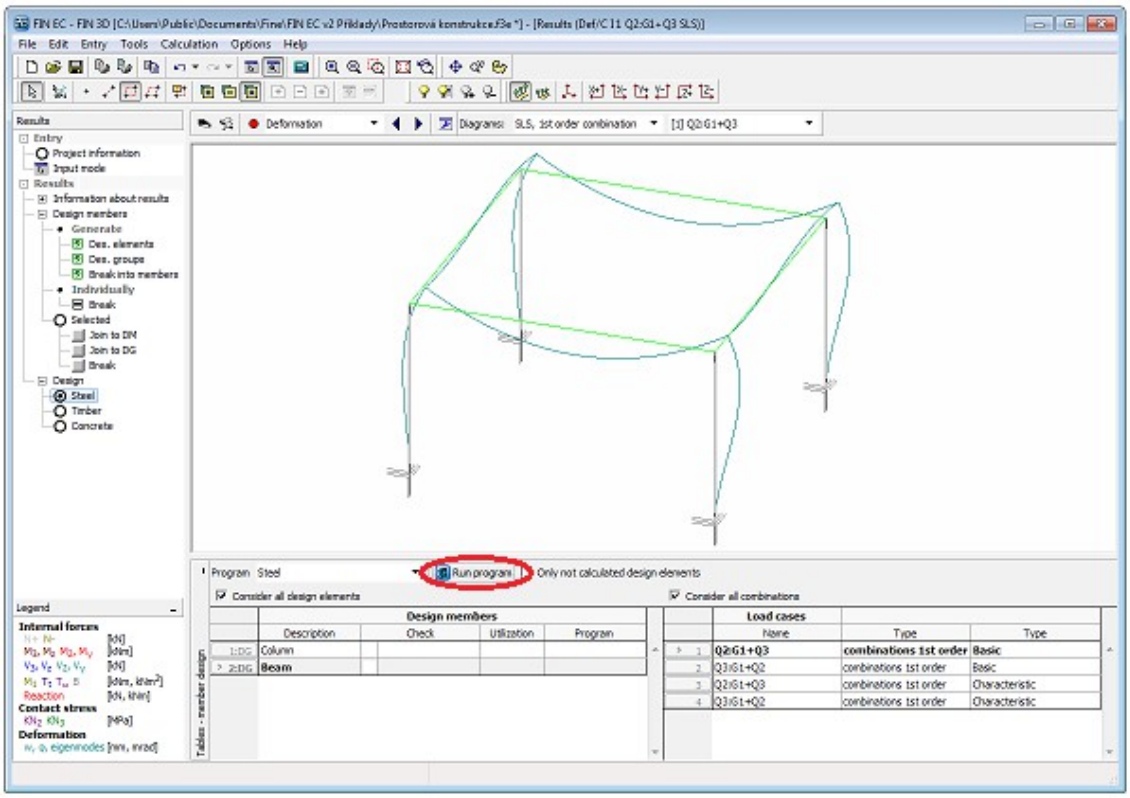
Notification of members' different orientation in a design group

After creating the design groups, we can name them in the "**Description**" column of the definition table in the bottom part of the main screen.



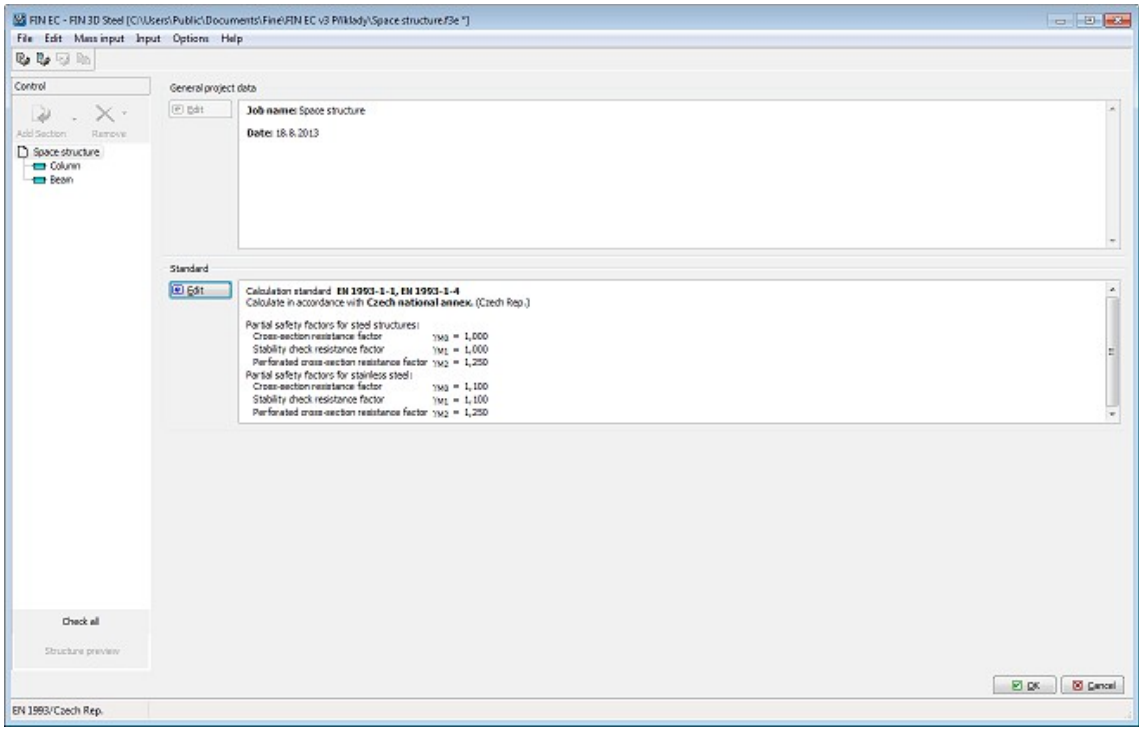
Naming design groups

Now we can carry out the design checks of the members by selecting "**Design**" and "**Steel**" in the control tree and clicking the "**Run program**" button for exporting data into the design program.



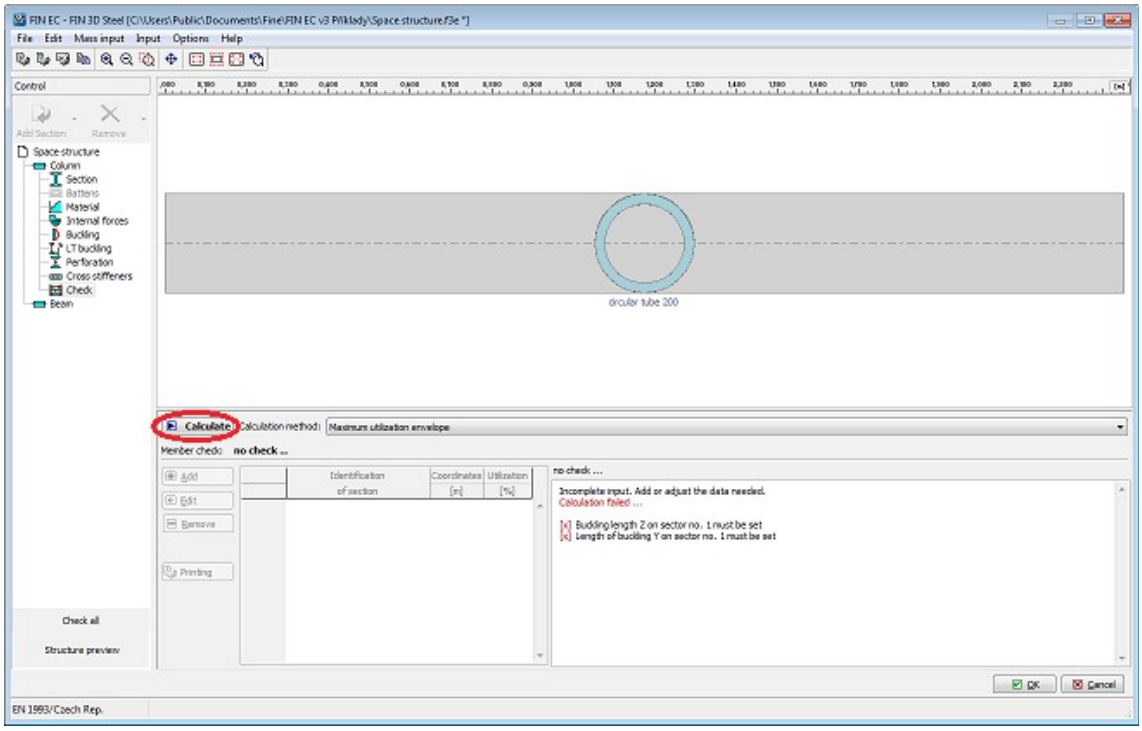
Running the design program

All necessary geometrical data (members' lengths, cross-sections, materials) as well as calculated internal forces are exported into the design program. The design groups are organized in the control tree on the left-hand side.


File Edit Mass input Input Options Help
Control
General project data
Job name: Space structure
Date: 18.9.2013
Space structure
Column
Beam
Standard
Edit
Calculation standard: EN 1993-1-1, EN 1993-1-4
Calculate in accordance with Czech national annex. (Czech Rep.)
Partial safety factors for steel structures:
Cross-section resistance factor γM0 = 1,000
Stability check resistance factor γM1 = 1,000
Perforated cross-section resistance factor γM2 = 1,250
Partial safety factors for stainless steel:
Cross-section resistance factor γM0 = 1,100
Stability check resistance factor γM1 = 1,100
Perforated cross-section resistance factor γM2 = 1,250
Check all
Structure preview
OK
Cancel
EN 1993/Czech Rep.

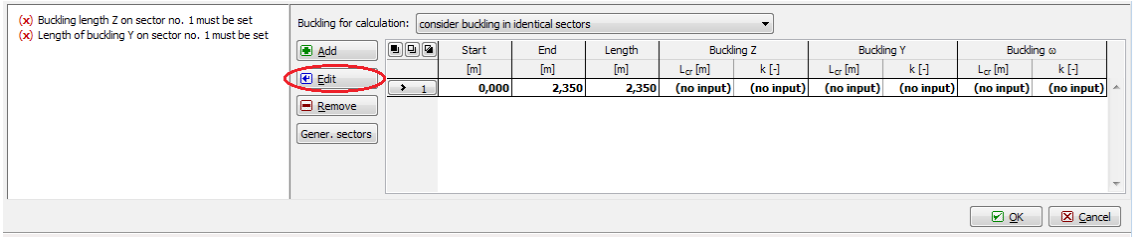
Design program for steel structures

First we carry out the design checks for the "**Column**" design group. We select "**Check**" in this design group's section of the control tree and click the button "**Calculate**" to run the members' analysis. Before the analysis starts, we are notified by the program that it is first necessary to set the buckling parameters.



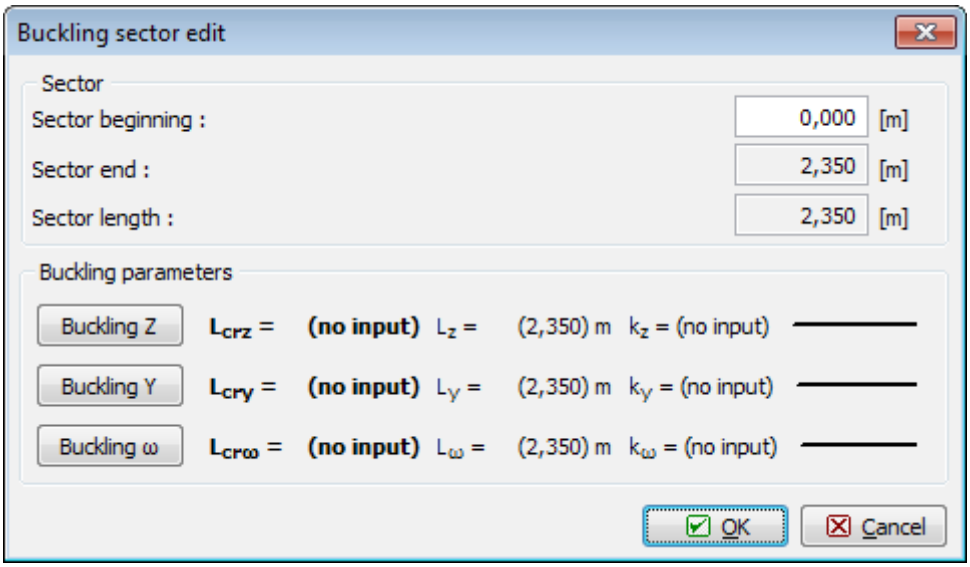
Members' design checks

Therefore we switch to the "**Buckling**" section of the control tree where we can set the required parameters by clicking the "**Edit**" button in the toolbar to the left from the definition table.



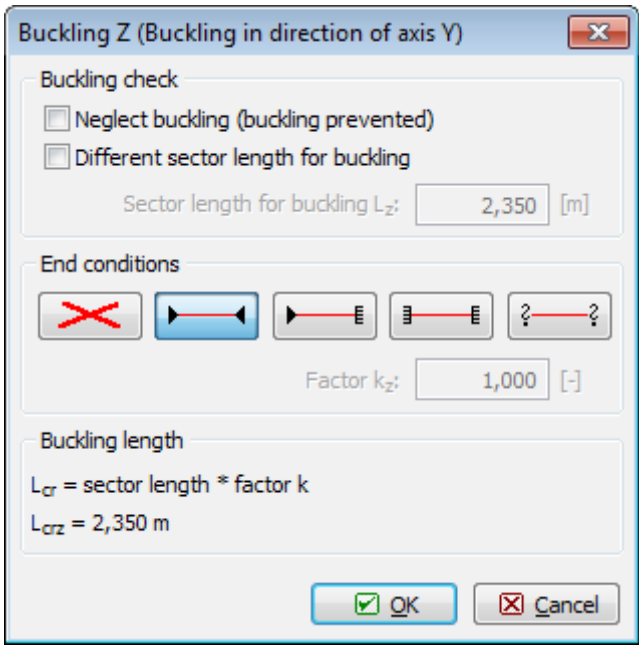
Button for editing buckling parameters

In the dialog box "Buckling sector edit" we need to set the buckling parameters in the z and y directions using the "**Buckling Z**" and "**Buckling Y**" buttons, where "**Buckling Z**" stands for displacement perpendicular to the Z-axis and "**Buckling Y**" analogically for displacement perpendicular to the Y-axis.



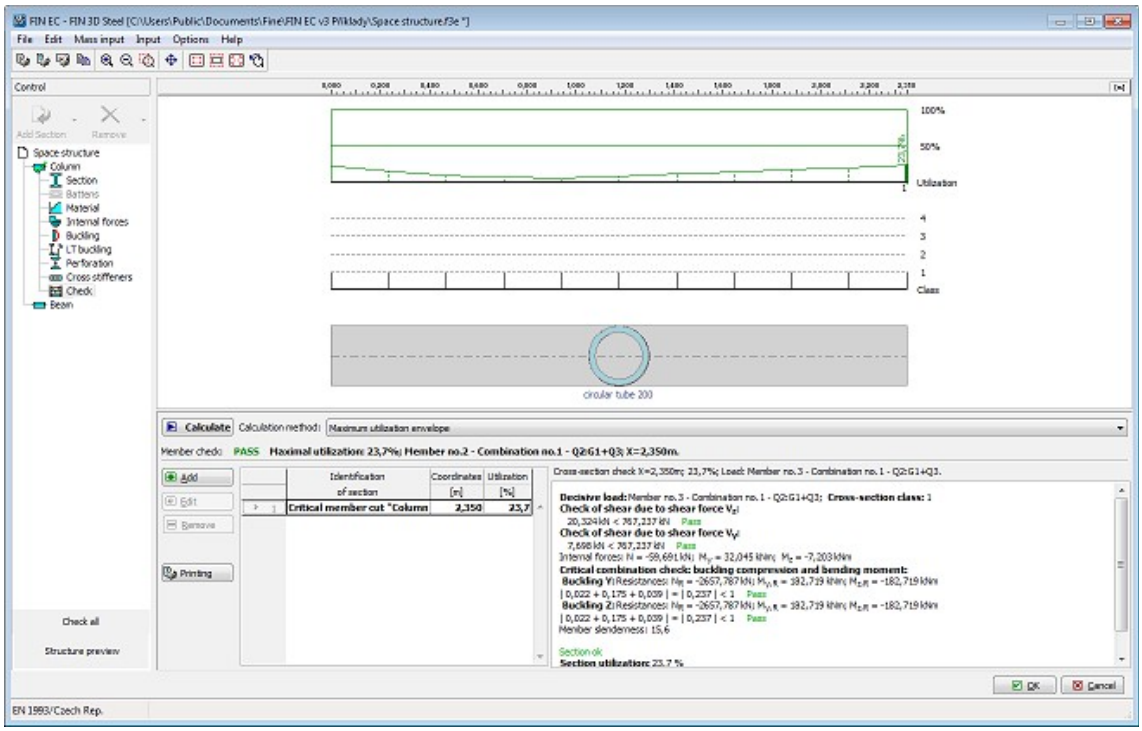
"Buckling sector edit" dialog box

In the "**Buckling Z/Y**" dialog boxes we need to select the end conditions. If necessary, we can also enter a buckling length different from the member length.
Apart from flexural buckling parameters, we normally need to define the lateral torsional buckling parameters in the section "**LTB**" - sector lengths, bending moment distribution types and end conditions.



"Buckling parameters"

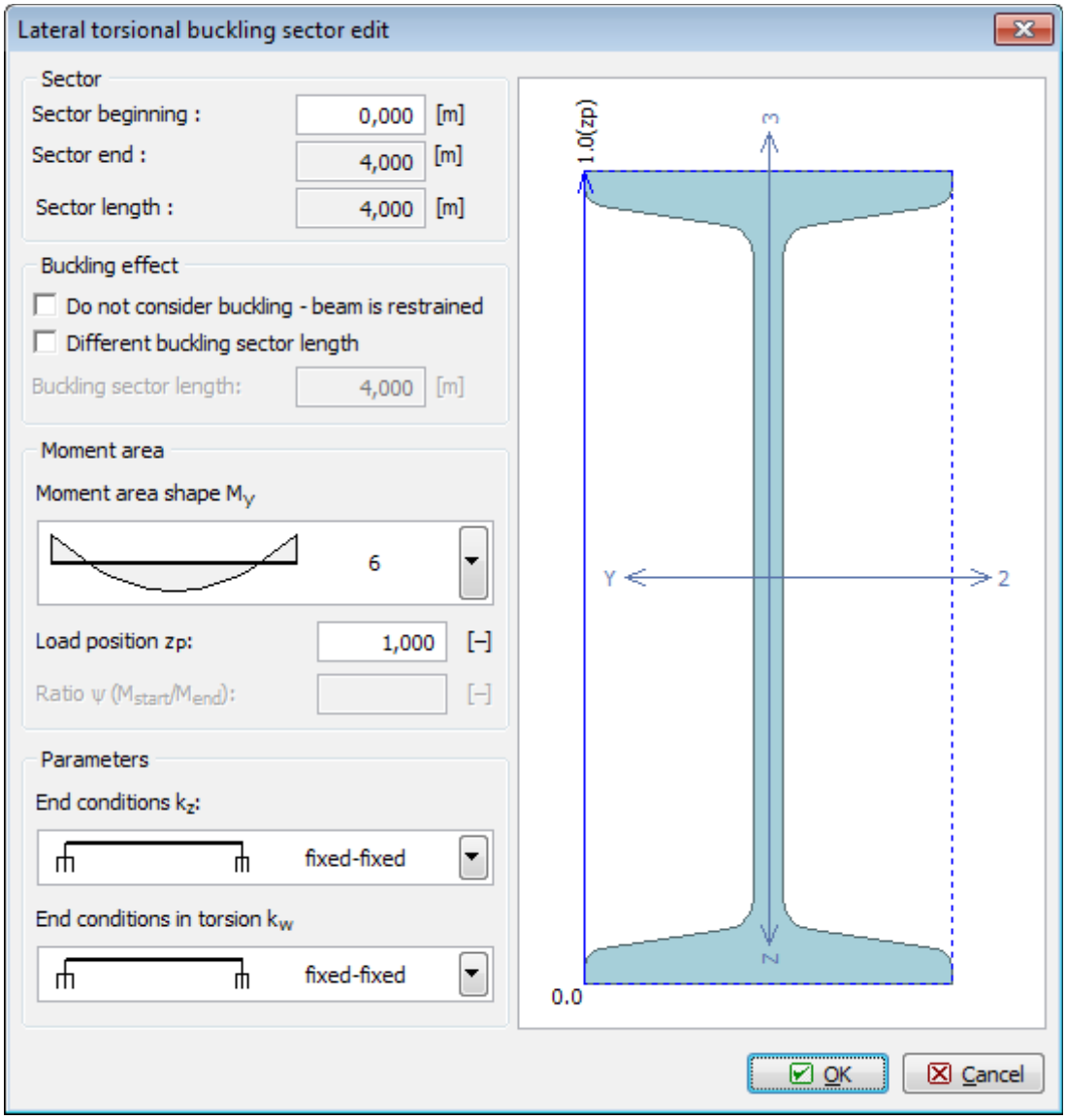
We define the buckling parameters analogically for both directions. Then we can switch back to "**Check**" and click the "**Calculate**" button. The design group containing columns passes all design checks.



Design checks of columns

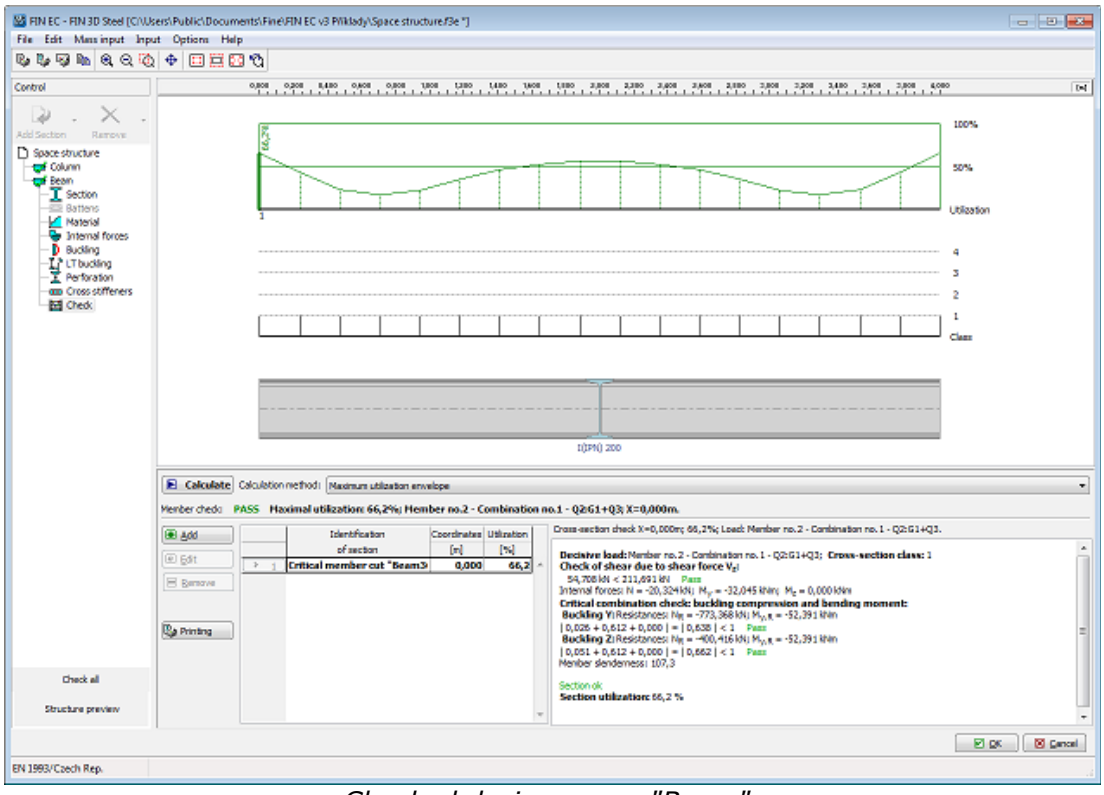
We proceed to checking the beams. As for the columns, we have to define the buckling parameters applying analogical procedure. However this time, with regard on the cross-section type, we need to define the lateral torsional buckling parameters from the bending moment M_y. The procedure is similar to definition of flexural buckling parameters. In the dialog box "**Lateral torsional buckling edit**" we need to select the moment area shape, load position in relation to the depth of the cross-section and also the end condition for flexure and torsion to

calculate the coefficients k_z and k_w.



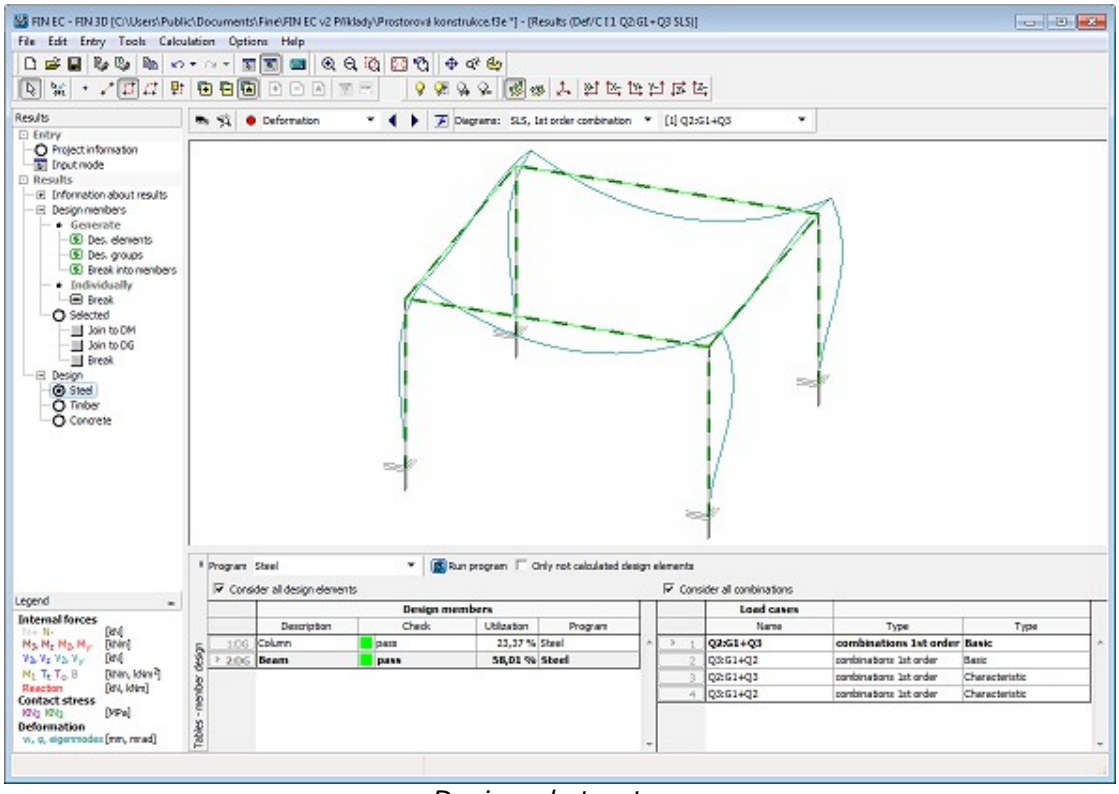
Dialog box "Lateral torsional buckling edit"

After defining the flexural and lateral torsional buckling parameters we can proceed to design checks.



Checked design group "Beam".

Members in both design groups pass all design checks. Therefore we can come back to the Fin 3D program window by clicking "**OK**". In the bottom part of the screen, the results of design checks (pass/fail) and maximum utilizations are summarized in a table.



Designed structure